

NASA Contractor Report 189153

1-53

Coupled Structural/Thermal/Electromagnetic Analysis/Tailoring of Graded Composite Structures

Final Status Report

(NASA-CR-189153) COUPLED
STRUCTURAL/THERMAL/ELECTROMAGNETIC
ANALYSIS/TAILORING OF GRADED COMPOSITE
STRUCTURES Final Status Report (GE) 53 D

N92-28296

Unclas
G3/39 0105610

M.S. Hartle, R.L. McKnight,
H. Huang, and R. Holt
General Electric Aircraft Engines
Cincinnati, Ohio

April 1992

Prepared for
Lewis Research Center
Under Contract NAS3-24538

NASA
National Aeronautics and
Space Administration

FOREWORD

This report has been prepared to expedite early dissemination of the information generated under the contract. The data and conclusions must be considered preliminary and subject to change as further progress is made on this program. This is a final report covering the work done during the 5 years of the contract. The NASA Program Manager is Dr. C.C. Chamis.

Table of Contents

Section	Page
1.0 INTRODUCTION	1
1.1 Executive Summary	4
2.0 TECHNICAL PROGRESS	11
2.1 Additional Capabilities	11
2.1.1 Buckling Capability	11
2.1.2 Mode Shape Slope Calculation	11
2.2 Program Verification	11
2.2.1 Stiffness Integration Study	12
2.2.2 Simulated Components	12
2.2.2.1 Exhaust Duct	12
2.2.2.2 Fan Frame	18
2.2.2.3 Fan Blade	20
2.2.2.4 Turbine Frame	31
2.2.2.5 Turbine Blade	34
2.3 Tailoring	34
2.3.1 Test Cases	34
2.3.1.1 Absorption Tailoring	34
2.3.1.2 Acoustic Tailoring	34
2.3.1.3 Frequency Tailoring	39
2.3.1.4 Weight Tailoring	39
2.3.1.5 Cost Tailoring	39
2.3.1.6 Thermal Tailoring	39
2.3.2 Simulated Components	41
2.3.2.1 Compressor Blade	41
2.3.2.2 Fan Blade	42
3.0 CONCLUSIONS	45

List of Illustrations

Figure	Page
1. Program Flowchart.	2
2. Composite Analysis System.	3
3. Total Composite Analysis System.	3
4. CSTEM System.	4
5. Constitutive Model – Structural Model Interaction.	5
6. CSTEM Analysis Module Flowchart.	9
7. CSTEM Tailoring Module Flowchart.	10
8. Frequency of Layer Integrated Isotropic Beam.	12
9. Coarse Mesh Duct.	14
10. Fine Mesh Duct.	14
11. Coarse Duct Axial Strain.	15
12. Fine Duct Axial Strain.	15
13. Coarse Duct Axial Stress.	16
14. Fine Duct Axial Stress.	16
15. Coarse Duct Material 1 Strain.	17
16. Fine Duct Material 1 Strain.	17
17. Fan Frame Loads.	18
18. Isotropic Fan Frame Deflected Shape.	19

List of Illustrations (Continued)

Figure	Page
19. Deflected Composite Fan Frame.	20
20. Representative Layups of Outer Case, Hub and Strut.	21
21. Composite Frame Material Properties.	22
22. Composite Fan Blade.	22
23. Mode 1, Predicted Frequency = 173.48 CPS.	23
24. Mode 2, Predicted Frequency = 506.72 CPS.	23
25. Mode 3, Predicted Frequency = 618.66 CPS.	24
26. Mode 4, Predicted Frequency = 1083.60 CPS.	24
27. Mode 5, Predicted Frequency = 1229.57 CPS.	25
28. Mode 6, Predicted Frequency = 1632.29 CPS.	25
29. Mode 7, Predicted Frequency = 1940.58 CPS.	26
30. Mode 8, Predicted Frequency = 2309.59 CPS.	26
31. Mode Shape Slope at 1341 RPM, Mode 1.	28
32. Mode Shape Slope at 1341 RPM, Mode 2.	28
33. Mode Shape Slope at 1341 RPM, Mode 3.	29
34. Mode Shape Slope at 1341 RPM, Mode 4.	29
35. Mode Shape Slope at 1341 RPM, Mode 5.	30
36. Mode Shape Slope at 1341 RPM, Mode 6.	30

List of Illustrations (Concluded)

Figure	Page
37. Simulated Turbine Rear Frame.	31
38. Temperature Distribution at 12 Seconds.	32
39. Temperature Distribution at 30 Seconds.	33
40. Deflected Shape at 30 Seconds.	33
41. CSTEM Versus THT-D and ANSYS Results.	35
42. CSTEM Heat Transfer Results for Turbine Blade Slice.	36
43. Simulated Turbine Blade Results (3, 2, 1).	36
44. Absorption Tailoring Model.	37
45. Absorption Tailoring Results.	37
46. Acoustic Tailoring Model.	38
47. Acoustic Tailoring Results.	38
48. Frequency Tailoring Model.	40
49. Frequency Tailoring Results.	40
50. Compressor Blade Model.	41
51. Coarse Mesh Composite Fan Blade.	42
52. Weight Tailoring of Fan Blade.	43
53. Tailored 1/2 Symmetric Layups.	44

List of Tables

Table	Page
1. Thermal Analyzer.	7
2. Layering Integration Error Results.	13
3. Composite Fan Blade Frequencies.	27

1.0 INTRODUCTION

This technical program is the work of the Life Assessment and Methods Technology Section of GE Aircraft Engines in response to NASA RFP 3-537260, CSTEM (Coupled Structural/Thermal/Electromagnetic) Analysis/Tailoring of Graded Composite Structures. The overall objective of this program is to develop and verify analysis and tailoring capability for graded composite engine structures taking into account the coupling constraints imposed by mechanical, thermal, acoustic, and electromagnetic loadings.

The first problem that was attacked is the development of finite elements capable of accurately simulating the structural/thermal/electromagnetic response of graded composite engine structures. Because of the wide diversity of engine structures and the magnitudes of the imposed loadings, the analysis of these is very difficult and demanding when they are composed of isotropic, homogeneous materials. The added complexity of directional properties which can vary significantly through the thickness of the structures will challenge the state of the art in finite element analysis. We are applying AE's 25 years of experience in developing and using structural analysis codes and the exceptional expertise of our University consultants toward the successful conclusion of this problem. To assist in this, we drew heavily on previously funded NASA programs.

We built on NASA programs NAS3-23698, 3D Inelastic Analysis Methods for Hot Section Components, and NAS3-23687, Component Specific Modeling, in the development of the plate and shell elements. In addition to these two programs, we drew on NAS3-22767, ESMOSS (Engine Structures Modeling Software System), and NAS3-23272, Burner Liner Thermal/Structural Load Modeling, in Task III when we generated a total CSTEM Analysis System around these finite elements. This guarantees that we are using the latest computer software technology and produced an economical, flexible, easy to use system.

In our development of a CSTEM tailoring system, we built on NASA Program NAS3-22525, STAEBL (Structural Tailoring of Engine Blades) and AE Program, AID (Automatic Improvement of Design) in addition to the program system philosophy of ESMOSS. Because of the large number of significant parameters and design constraints, this tailoring system will be invaluable in promoting the use of graded composite structures.

All during this program, we availed ourselves of the experience and advice of our Low Observables Technology Group. Their input was used to assure the relevance of the total program.

Figure 1 shows our program and major contributions in flowchart form. This gives a visual presentation to the synergism that exists between this program and other activities.

Figure 2 depicts an integrated analysis of composite structures currently under development in the composite users' community. The severe limitations of such a system are not highlighted because three major steps in the process are not shown. Figure 3 adds these steps. The analysis system really begins with a definition of geometry. A user then defines a finite element model simulating this geometry and the anticipated loading. The process then moves to defined Step 3. One cycle through the process ends with the prediction of individual ply average stresses and strains. Now comes a significant productivity drain, namely, manual intervention to evaluate these stresses and strains against strength and durability limits. Based on this, the user must decide to (1) change the finite element model, (2) change the composite laminate, (3) both of the above, or (4) stop here.

Obviously, there is a considerable cost savings to be obtained by selecting Number 4. The CSTEM system will obviate the reasons for selecting Number 4. This system, shown in Figure 4,

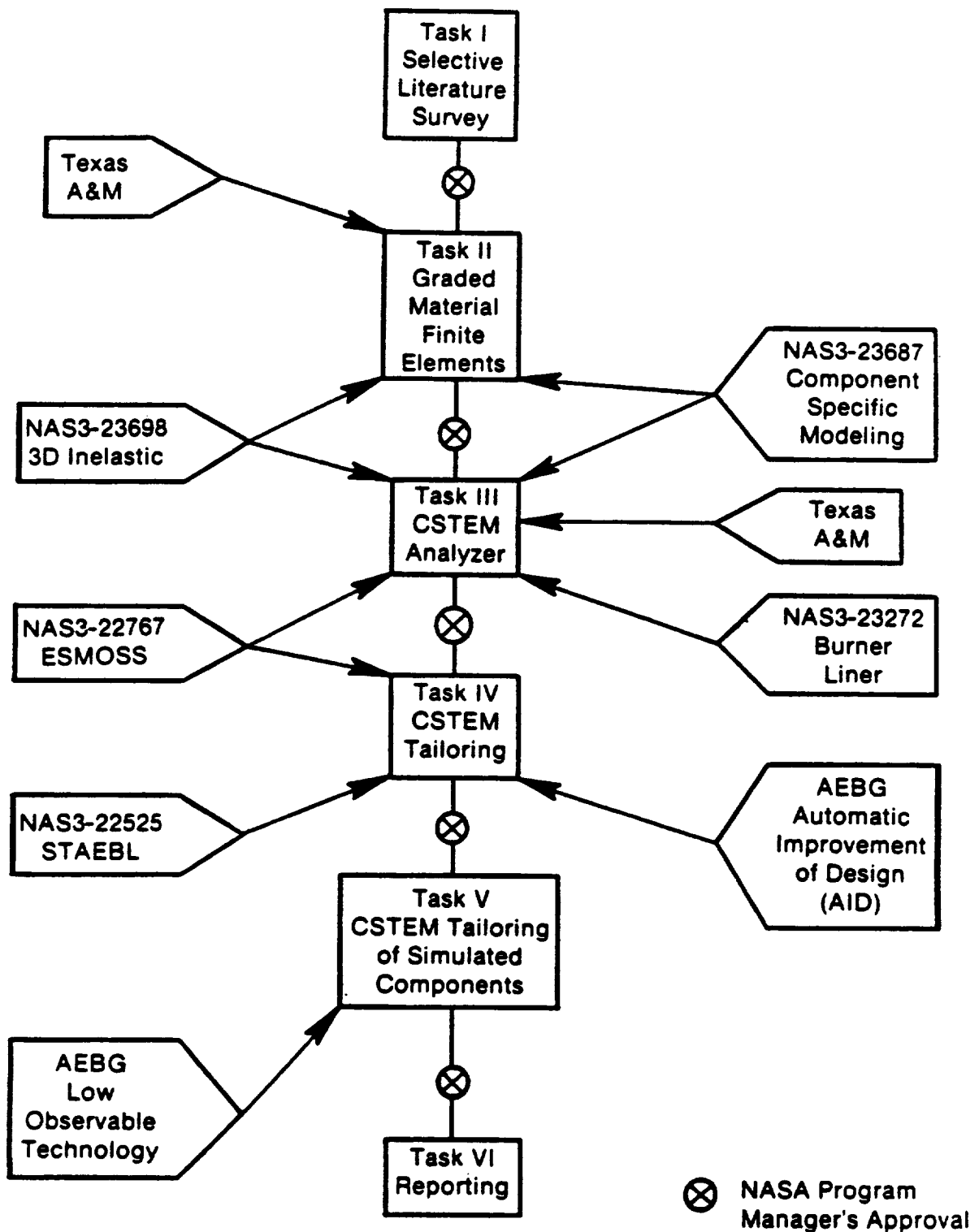


Figure 1. Program Flowchart.

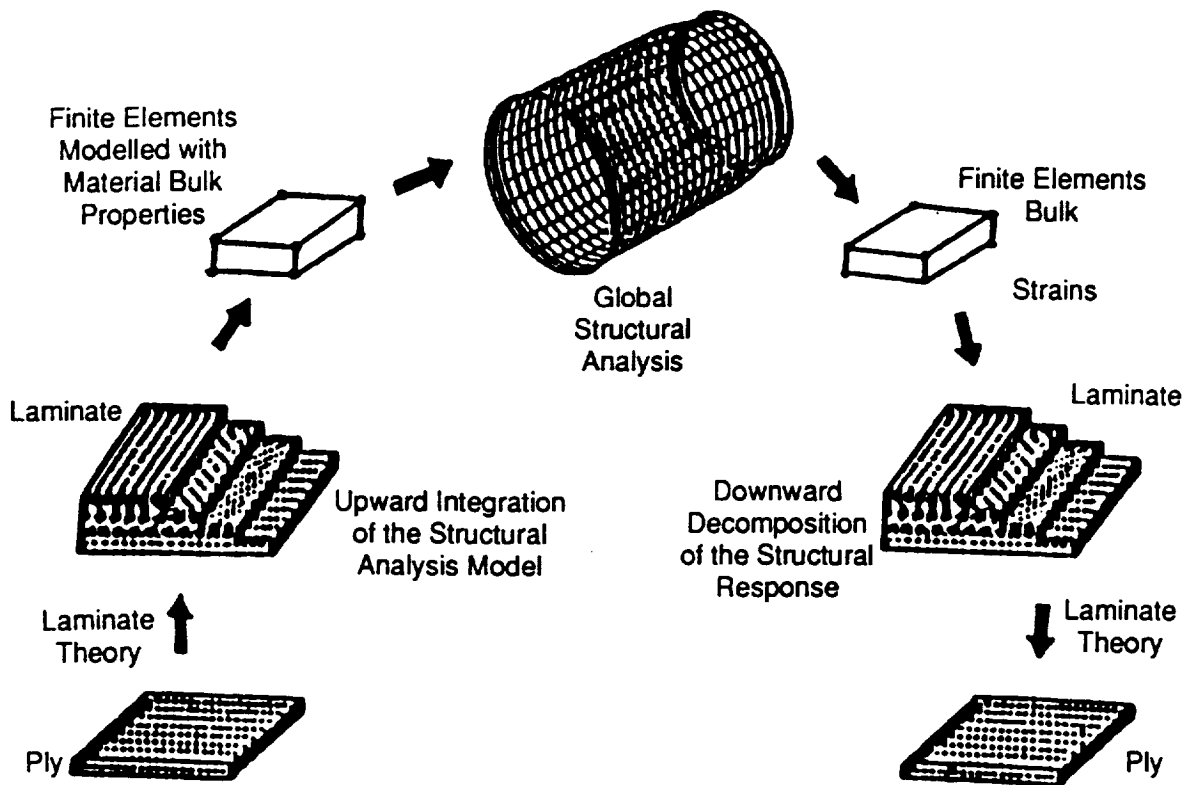


Figure 2. Composite Analysis System.

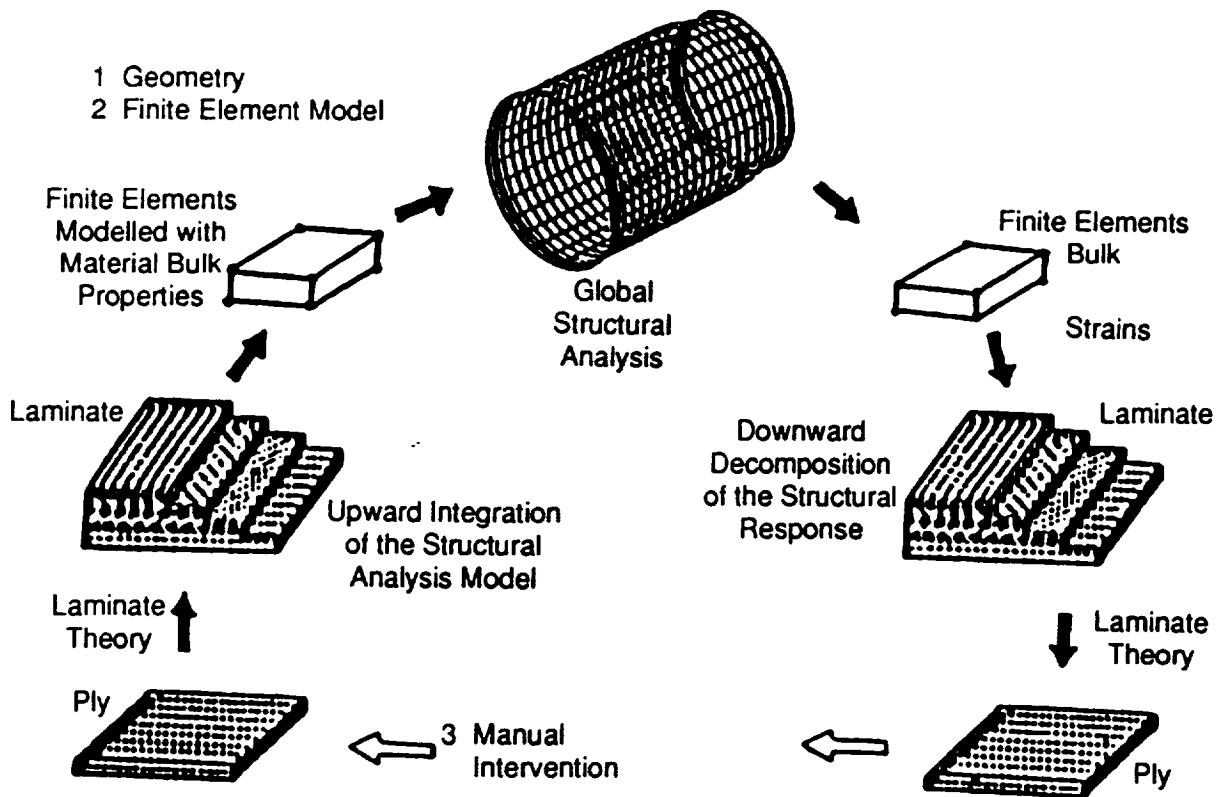


Figure 3. Total Composite Analysis System.

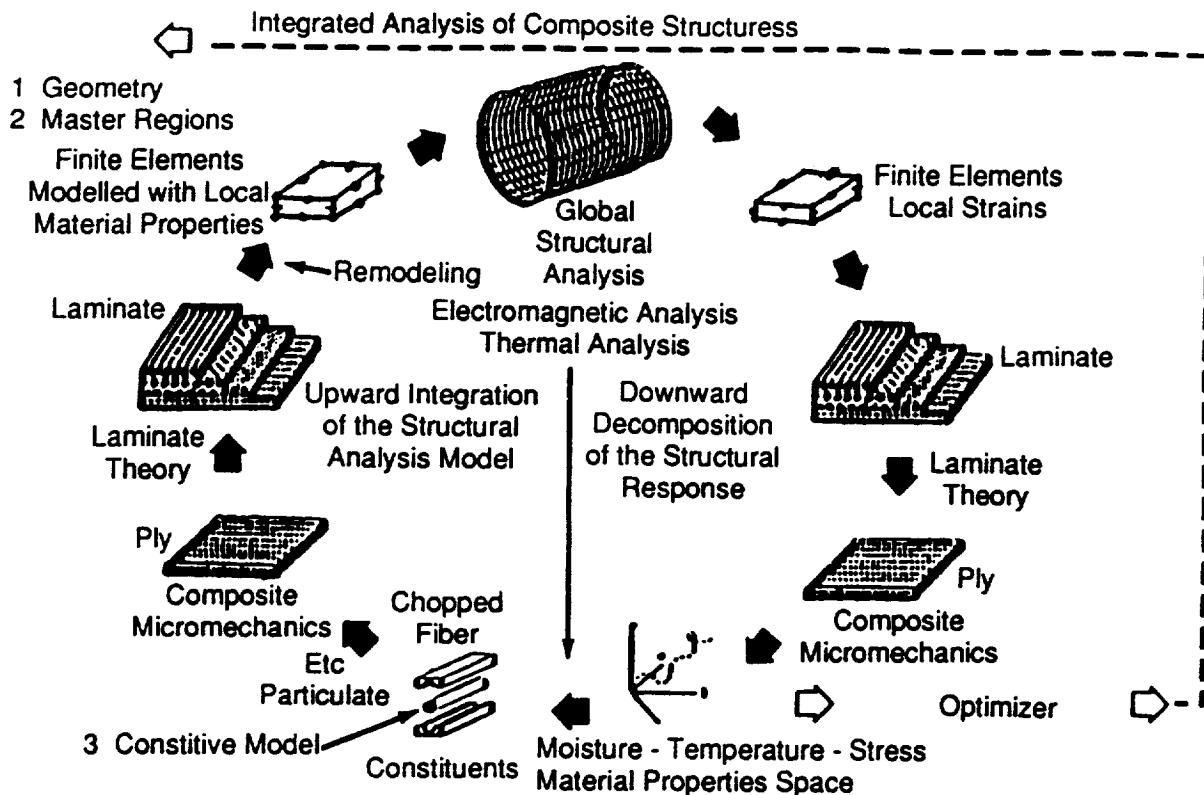


Figure 4. CSTEM System.

begins with the definition of geometry, as before, but then proceeds to a definition of master regions which contain all of the necessary information about geometry, loading, and material properties. Step 3 is a constitutive model which develops the necessary structural, thermal, and electromagnetic properties based on a micromechanics approach. Furthermore, this constitutive model contains the logic to generate the global finite element model based on the variation of the properties, as depicted in Figure 5. Using a nonlinear incremental technique, those global models are solved for their structural, thermal, and electromagnetic response. Based on this response the global characteristics are evaluated, with convergence criteria and decisions made on remodeling. Once the global characteristics meet the accuracy requirements, the local characteristics are interrogated and decisions made on remodeling because of strength, durability, or hereditary effects. Once this cycle has been stabilized, optimization is performed based on design constraint.

1.1 Executive Summary

"CSTEM" is the acronym for the computer program being developed under the NASA contract, "Coupled Structural/Thermal/Electromagnetic Analysis/Tailoring of Graded Composite Structures." The technical objectives for this program are to produce radar signal transparent structures having high structural performance and low cost. The multidisciplines involved are all highly nonlinear. They include anisotropic, large deformation structural analysis, anisotropic thermal analysis, anisotropic electromagnetic analysis, acoustics, and coupled discipline tailoring. The CSTEM system is a computerized multidiscipline simulation specialized to the design problems of radar absorbing structures. The enabling technical capabilities are implemented in a

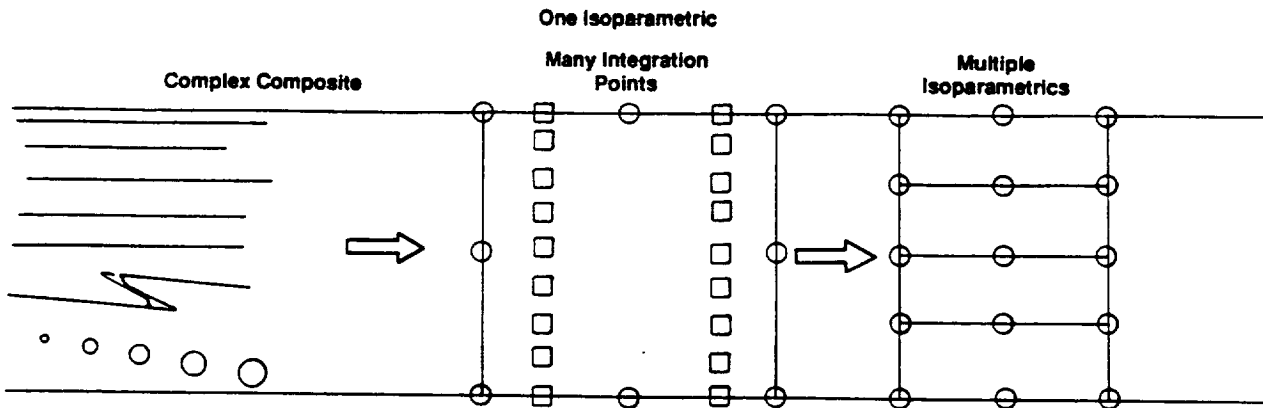


Figure 5. Constitutive Model – Structural Model Interaction.

special 3D finite element formulated to simultaneously tailor the geometrical, material, loading, and environment complexities of radar transparent structures for cost effective optimum performance.

In each enabling technical discipline a decoupled stand-alone 3D finite element code has been developed. An executive program with controlling iterative solution techniques performs the nonlinear coupling among the participating analysis modules. Each analysis module is self contained, passing only the required input geometry and control information between the modules as well as returning any results which may be required as input for an analysis by a following module.

The structural module uses 8, 16, or 20 noded isoparametric bricks and is similar to many other isoparametric finite element codes in many ways. It has the capability for centrifugal, acceleration, nodal displacement, nodal force, temperature, and pressure loadings. The solution technique used is a multiblock column solver, which allows solution of very large problems since it can work on portions of the set of equations separately.

The more advanced features of the structural module include its orthotropic material capability. Material properties can be input relative to the material axes and then skewed on an element by element basis to obtain the desired orientation of the material with the global coordinate system. Material properties may also be referenced to the elemental coordinate system, with the orientation between elemental and global being calculated internally. The structural module can also generate the orthotropic material properties it needs for composite materials using the constituent properties making up the composite. This is done using an internal adapted version of the computer program INHYD, which accesses a data bank containing the material properties of the constituents. The properties are calculated based on the volume ratios of the constituents.

Another advanced feature of the CSTEM structural module is its multiple layer capability. This allows the modeling of composite structures with many material layers without the necessity of using an element for each layer. The stiffness of an element with multiple layers is calculated using integration points located on the midplane of each layer within the element. The composite stiffness gradient controls the finite element definition of a structure with two major parameters to vary: the number of elements through the gradient and the number of numerical quadrature points within an element. A unique set of local stiffness characteristics is developed for each numerical integration point. Integration of these local characteristics over the volume of the element provides total element simulation of composite structures, including such effects as twist-bend coupling. The stress and strain are then recovered at these same integration points.

The structural analyzer also performs large deformation analysis using a unique incremental updated Lagrangian approach with iterative refinement. Testing of this capability against classical large deformation problems has shown it to be both more accurate and more economical than available alternatives. Connected with this capability is a deformed position eigen-analysis capability. All or selected portions of the nonlinear stiffness terms can be incorporated into these eigen-analyses. This capability has been checked out against available test data and other computer codes.

There is a requirement that the element shape follows the layup of the structure so that the layers cut through opposite faces of the element at the same height and not diagonally across the element. This requirement points to the use of a mesh generator, which is a part of the CSTEM structural module. The CSTEM mesh generator is capable of producing various solids of revolution from a minimum of input parameters. The generator can produce flat surfaces, cylinders, cones, and general double curved surfaces of up to 360° rotation. These different surface types can be generated together. The generator will check for coincident nodes and keep only one of any nodes which have the same coordinates. When generating more than one surface, care must be taken that the surfaces are generated so that the connecting nodes will have the same coordinates.

Another capability that can be used together with the multiple layer capability is a composites analyzer, which is adapted from the ICAN computer program. This capability must also be used together with the INHYD generation of material properties. The composites analyzer takes the stress/strain results from the structural module and integrates them through the thickness of the structure at some user specified location. This results in a loading which can be used by the composites analyzer to do a microanalysis of the composite at that particular location.

The heat transfer module has the capability to perform four different analysis types. In increasing order of complexity these are linear steady state, nonlinear steady state, linear transient, and nonlinear transient. The types of loadings that can be used in the heat transfer analyses are nodal and surface heat fluxes, convection, radiation, and internal heat generation. The material thermal conductivity and specific heat are required as material properties. Table 1 lists the parameters involved in the various heat transfer analysis types.

In the nonlinear analyses, material properties are entered at several temperature points and interpolated to the calculated temperatures. The solution is iterated upon until convergence is achieved.

In the transient analyses, time steps are specified and boundary conditions are entered at each of these time steps. The solution is obtained and printed out at each time step point by stepping along a series of evenly spaced user specified time subincrements.

When doing the heat transfer analysis as part of a coupled solution, the calculated nodal temperatures are passed to the structural module so that the structural material properties and thermal strains are calculated using these temperatures. A structural solution can be obtained at specific time step points as requested by the user input in the case of a transient heat transfer analysis.

The layering capability of the structural module is also used in heat transfer. This and the ability to specify orthotropic material thermal conductivities provides the capability to perform heat transfer analyses of composite materials. The INHYD micromechanics program for generating laminate material properties from the material constituents is also available from the heat transfer module to generate thermal properties.

Table 1. Thermal Analyzer.

Thermal Parameters and Boundary Conditions	Steady State		Transient	
	Linear	Nonlinear	Linear	Nonlinear
• Temperature	T	T	$T(t)$	$T(t)$
• Time	---	---	t	t
• Thermal Conductivity	k_{ij}	$k_{ij}(T)$	$k_{ij}(t)$	$k_{ij}(T,t)$
• Convection Coefficient	h	$h(T)$	$h(t)$	$h(T,t)$
• Internal Heat Generation	Q_i	Q_i	$Q_i(t)$	$Q_i(t)$
• Surface Heat Flux	Q_s	Q_s	$Q_s(t)$	$Q_s(t)$
• Convection Boundary	Q_c	Q_c	$Q_c(t)$	$Q_c(t)$
• Specified Nodal Temperatures	T_s	T_s	$T_s(t)$	$T_s(t)$
• Heat Capacity	---	$C_p(T)$	---	$C_p(T,t)$
• Radiation Emissivity	---	$\epsilon(T)$	---	$\epsilon(T,t)$
• Viewing Factor	---	f	---	$f(t)$

The electromagnetic absorption module has three options for calculation of absorption. All three methods use a data bank of absorption material properties. The first method uses the computer program WAVES as a subroutine in CSTEM. This program calculates the reflection and transmission of electromagnetic waves given a layup sequence of materials and their electromagnetic properties. Using this layup the WAVES program develops complex impedances to calculate reflection and transmission coefficients for the cross section. The electromagnetic properties needed are complex values of the permittivity and permeability, which are obtained from the data bank.

The second method calculates reflection, refraction, and attenuation of electromagnetic waves by using Snell's Law, the Fresnel Formulas, and the attenuation constant as derived from the vector wave equations. This method requires the material properties of permittivity, permeability, and conductivity to be available as a function of temperature and frequency on a data bank similar to the first method.

The third method uses a data bank that is different from the first two methods in that it contains absorption properties for the material at not only discrete values of temperature and frequency, but also polarization angle. The absorption of electromagnetic energy of a specific frequency and polarization by a given material at a specific temperature is calculated by linearly interpolating from the discrete data bank values.

The orientation of an electromagnetic wave is specified similar to a skew material so that a coordinate system is associated with the wave propagation. This wave coordinate system is defined such that the direction of propagation is along the positive Z axis and polarization is measured from

the positive X axis. The orientation of the wave coordinate system with the global coordinate system is specified using skew transformations.

The element face upon which the electromagnetic wave is impinging is specified by the input. The path taken through the structure thickness is determined by the program assuming that the wave always exits through the opposite element face that it entered. Absorption calculations are made for each material encountered and are carried out using midsurface centroid values of temperature and orientation. The impingement angle is calculated as a dot product of the wave coordinate system Z axis and the midsurface centroid normal. The polarization angle is calculated as the dot product of the projection on the layer midsurface of the wave polarization and the material orientation,

Absorption calculations are done for one given frequency, orientation, and wavepath at a time. If it is necessary to calculate results for several frequencies, orientations, or wavepaths, a separate calculation must be done for each combination.

The approach to calculate acoustic characteristics due to structural vibration in CSTEM determines the radiation efficiencies of a structure for each vibration mode as a function of frequency. An eigen-analysis produces the fundamental modes and mode shapes. Once the radiation efficiencies for each mode are calculated, the total sound power is obtained by a modal summation of the contribution from each mode.

CSTEM tailoring capability has been built on the STAEBL computer program obtained from NASA Lewis. This program consists of two major modules: CONMIN, which performs the actual tailoring, and ANALIZ which supplies the parameters to be tailored. The CONMIN module was abstracted from STAEBL and coupled with the CSTEM structural, thermal, electromagnetic, and acoustic analysis modules.

Figure 6 contains a flowchart of the major analysis modules of CSTEM. These modules are used as a stand alone analysis package with entry through the main executive routine, or as the analysis portion of the tailoring process in which case the entry to these modules is at the load case level. Figure 7 is the flowchart of the tailoring process itself.

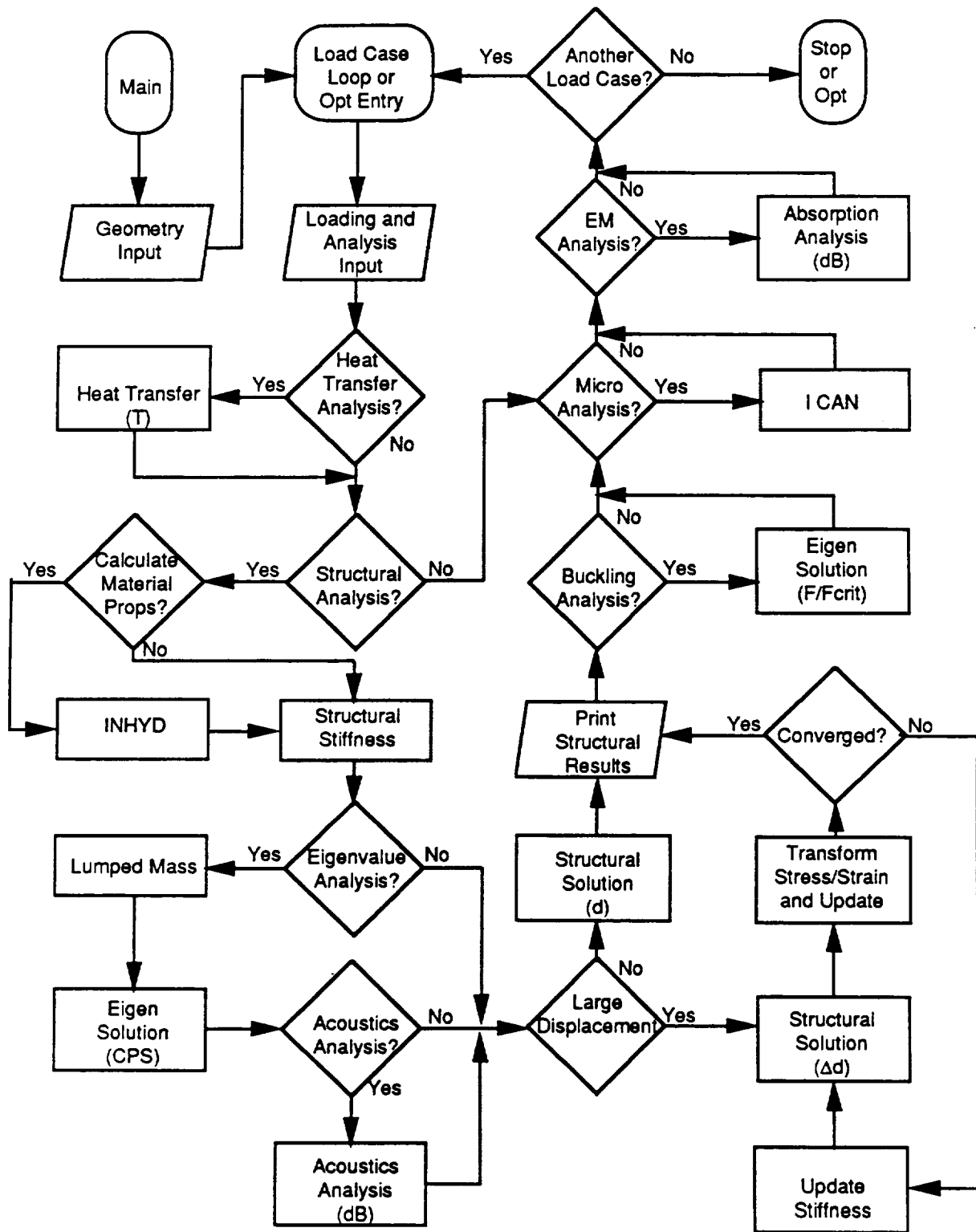


Figure 6. CSTEM Analysis Module Flowchart.

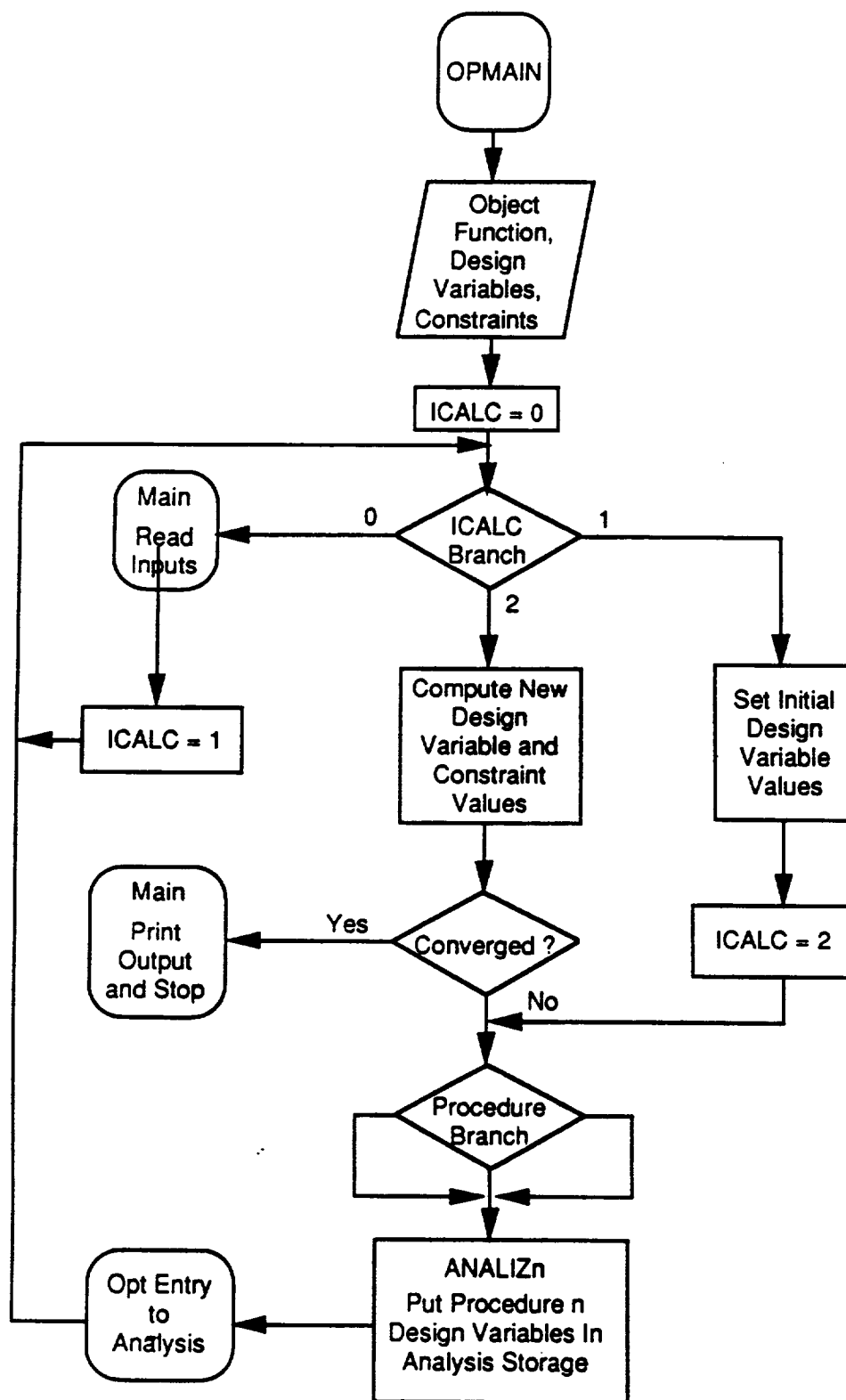


Figure 7. Tailoring Process Flowchart.

2.0 Technical Program

2.1 Additional Capabilities

2.1.1 Buckling Capability

Buckling capability was added to CSTEM by combining features available from other capabilities of the code. The buckling analysis is very similar to a free vibration analysis with the negative of the stress stiffness matrix used in place of the mass matrix. The resulting eigenvalues are multipliers to the current load that would give the critical load. This means that an eigenvalue of 1.0 indicates that the load is at the critical value. A negative eigenvalue indicates that buckling cannot occur in the current load configuration. The inherent assumption in using the eigenvalues as load multipliers is that the load to stiffness relationship is linear.

Buckling analysis can be turned on and off from load case to load case. Stiffnesses relating to the original configuration or to the current displaced configuration can be used allowing geometric nonlinearities to be taken into account.

Checkout cases on columns in which the critical Euler loads can be easily calculated give excellent results. For a 10 inch long fixed end column with a 1 x 1 inch square cross section and a Young's modulus of 10E6 the Euler load is 20561 pounds. Modeling this column with a 1000 pound compressive end load gives an eigenvalue of 20.9 with multiplicity of 2. For a 1 x 2 inch cross section the Euler load in the short dimension is 41123 and 164493 in the long dimension. The resulting eigenvalues for this configuration were 42.1 and 163.3 using the same 1000 pound end load. Increasing this load resulted in a similar decrease in the eigenvalues, with an extreme of a 100000 pound load on the 1 x 1 column resulting in eigenvalues of 0.21. Using a tensile load, the eigenvalues become negative and convergence was not achieved in the 16 iteration limit.

2.1.2 Modeshape Slope Calculation

The maximum value of the modeshape slope is considered an important criterion in the design of airfoils, particularly in relation to stability. A routine (MSHPSLOP) that approximates the chordwise slope of a surface represented by the eigenvectors of up to 8 of the lowest natural frequencies has been implemented in CSTEM. This approximation is computed as described below.

The nodes of the finite element model are points on the surface of an airfoil on sectors from the root to the tip. The Lagrangian surface defined by 3 consecutive sectors establishes a coordinate plane. The cross partial derivative of the x and y components of the eigenvectors is defined as the modeshape slope at the point on the chordline. This derivative is computed by 3 point Lagrangian interpolation.

2.2 Program Verification

A large part of the effort of the past year has been on verification of the CSTEM program. Although many test cases have been run as various capabilities were added to the program, a verification of the complete package is necessary. This was done by analyzing simulated components and examining the results to determine whether they are reasonable. In some cases, actual components were used and results from previous analyses using other codes could be compared to those obtained from the CSTEM code.

2.2.1 Stiffness Integration Study

The layered 3D element in CSTEM is based on the assumption that the calculation of the element stiffness can be fairly accurately done using a Gaussian distribution on the layer midplanes and summing over the layers in the third direction. A study of the error involved in using this method to integrate element stiffnesses was done. The free vibration frequency of an isotropic cantilever beam was calculated using several different numbers of equal thickness layers. Figure 8 shows the results of this study in comparison to using a 3x3x3 Gaussian integration, which would give the best expected result for the 20 noded brick elements. It can be seen that even with 3 layers the results are within 6% of those obtained with the exact integration. Table 2 shows some additional results for uneven thickness layers. It can be deduced from this study that any reasonably close to even thicknesses will give good results; however, extreme differences in thicknesses can give erroneous results.

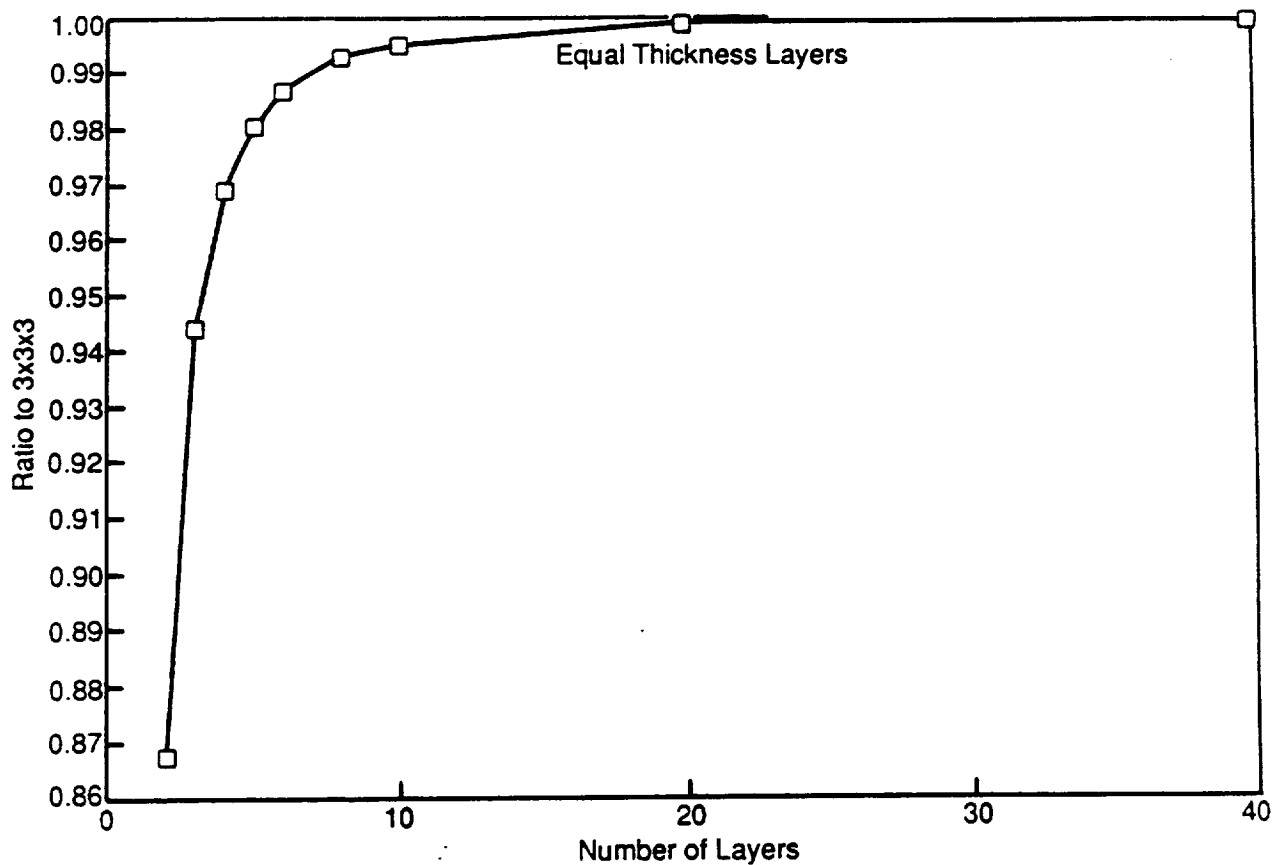


Figure 8. Frequency of Layer Integrated Isotropic Beam.

2.2.2 Simulated Components

2.2.2.1 Exhaust Duct

A simulated exhaust duct was analyzed using different mesh densities to examine the sensitivity of the solutions in terms of the mesh density. When using the tailoring capability of the CSTEM

Table 2. Layering Integration Error Results.

No. Of Layers	Layer Distribution					Ratio To $3 \times 3 \times 3$
2	60%,	40%				0.85
	20%,	80%				0.69
3	20%,	60%,	20%			0.88
	10%,	80%,	10%			0.70
	2%,	96%,	2%			0.34
4	20%,	30%,	30%,	20%		0.96
	10%,	40%,	40%,	10%		0.93
	2%,	48%,	48%,	2%		0.88
5	10%,	27%,	26%,	27%,	10%	0.97
	2%,	32%,	32%,	32%,	2%	0.95
	1%,	1%,	96%,	1%,	1%	0.34
6	10%,	4*(20%),	10%			0.98

program several solutions of a problem may be needed to define an optimum of parameters. It is therefore necessary to use a coarse model of the component to reduce execution time.

The simulated duct is 20 inches long, has a 10 inch I.D. and a 1 inch wall thickness. Two mesh densities were used, the coarse mesh containing 264 nodes and 34 elements, and the fine mesh containing 752 nodes and 96 elements. The meshes are shown in Figures 9 and 10. The duct is composed of 4 layers through the wall thickness. Each layer is the same material, T300/IMHS from the ICAN data bank. The layers are oriented at +60, -60, -60, +60 with respect to the duct circumferential direction. The duct is fixed at the root end and is loaded with thermal and pressure loads, with the major loading being a tip displacement of 1 inch in the X direction. A 3 by 3 Gauss integration is used for each layer in an element.

The results presented are from three integration points per element, all located at the same axial position in the respective models. However, since the fine mesh model was refined axially as well as circumferentially the integration points are not at the same axial locations as in the coarse mesh model. The axial location of the displayed results of the coarse mesh model is 5 inches from the root while the fine mesh results are for an axial location of 3.33 inches from the root. This prevents a strict quantitative comparison between the models, but a qualitative comparison is still illuminating. Figures 11 and 12 compare the axial (global Y) strain about the circumference for the coarse mesh and fine mesh as calculated in the root elements. Figures 13 and 14 compare the axial stress for the two models. Figures 15 and 16 compare the strains in the fiber direction of each ply for the two models.

The results from this examination tend to verify the concept that should be applied in tailoring components using CSTEM. The effect of the mesh coarseness is apparent in the plots, especially between elements. When looking at trends however, the coarse mesh predicts maximum and minimum occurrences at the same locations as the finer mesh. The actual maximum and minimum predicted values also appear to be close. The global strain values are very continuous as expected. The global stress values show a distribution that depends on the strains in the fiber directions. This

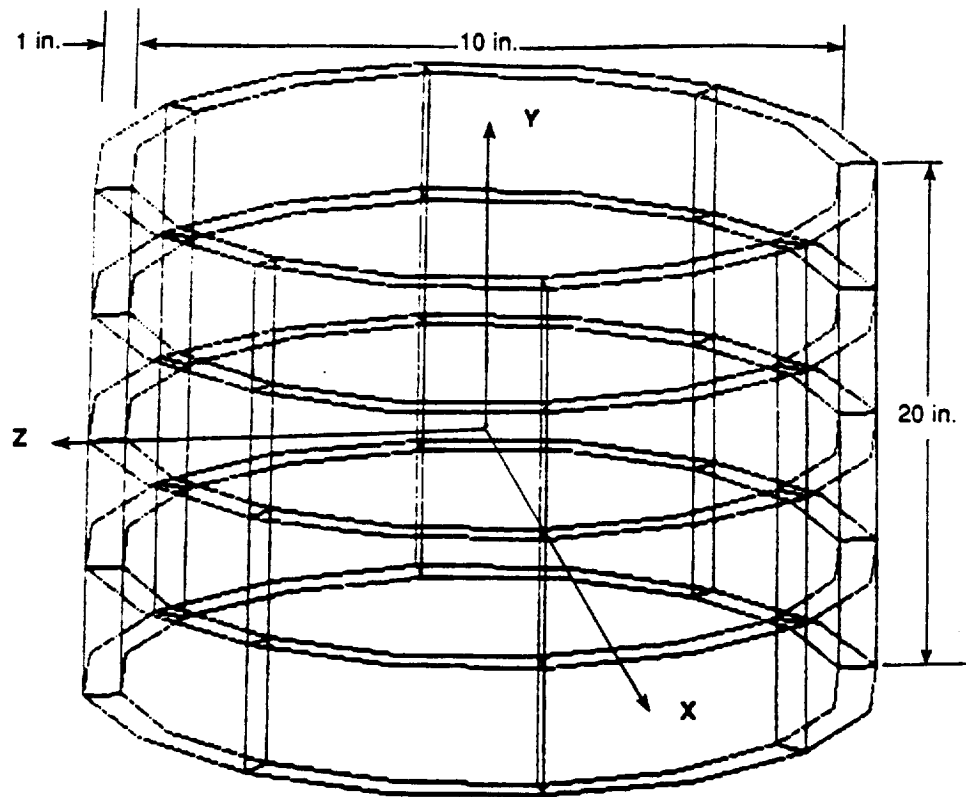


Figure 9. Coarse Mesh Duct.

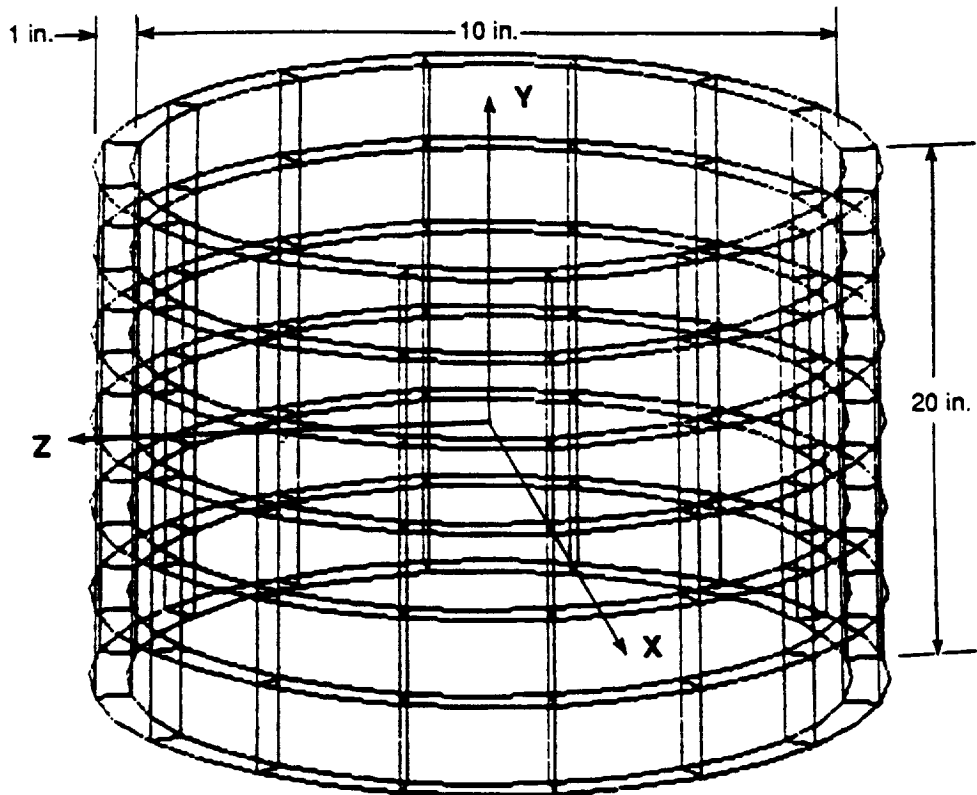


Figure 10. Fine Mesh Duct.

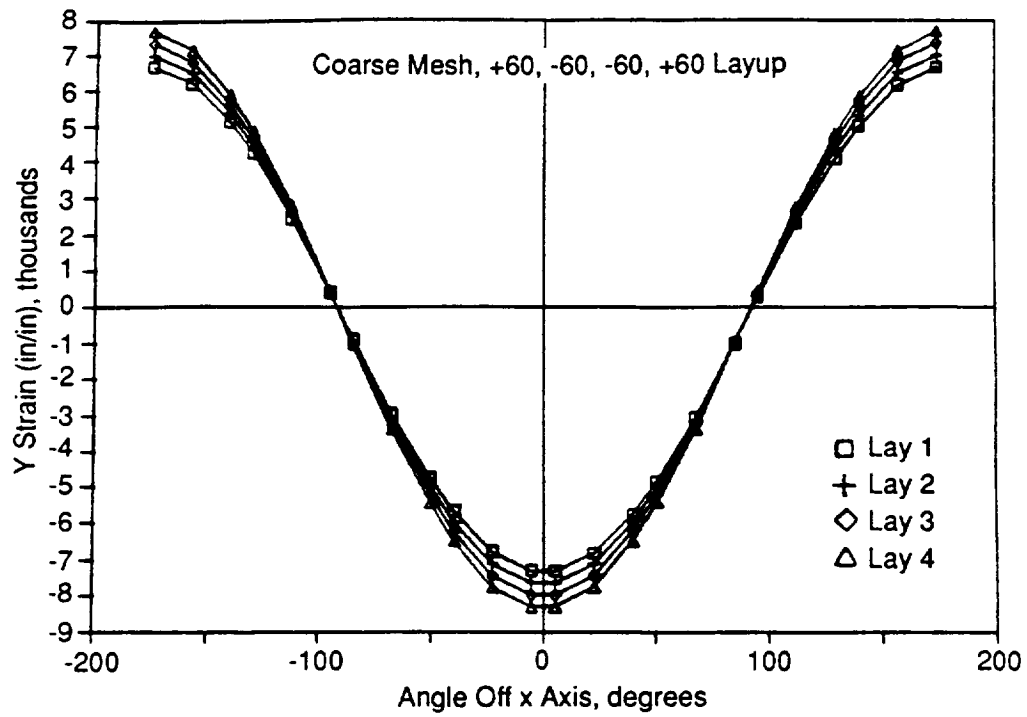


Figure 11. Coarse Duct Axial Strain.

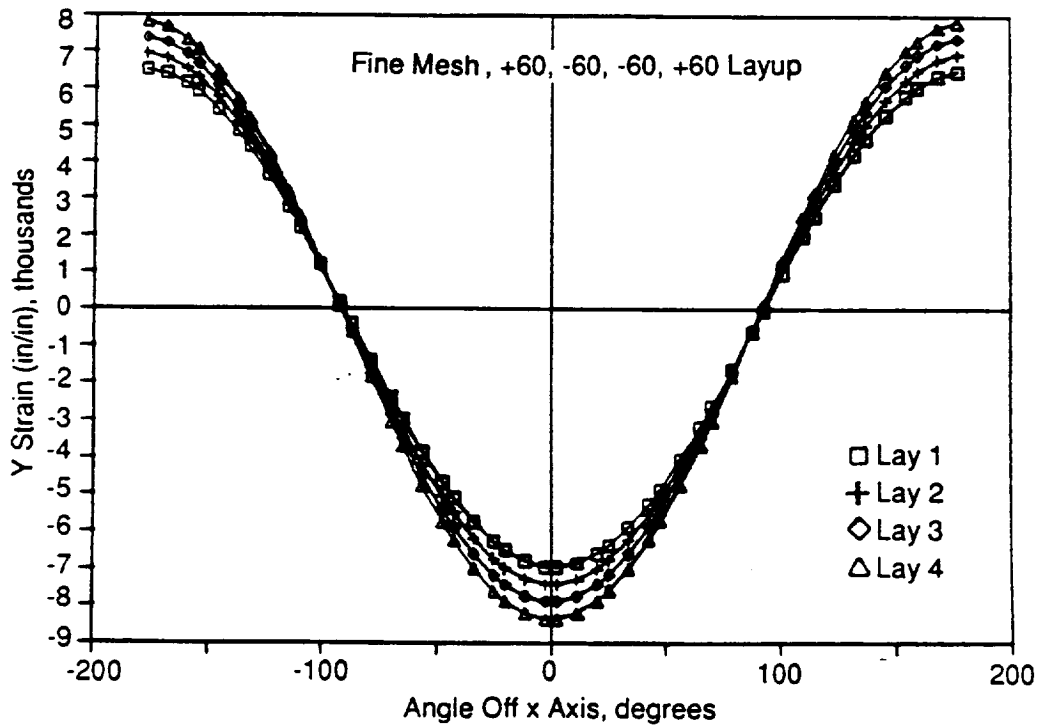


Figure 12. Fine Duct Axial Strain.

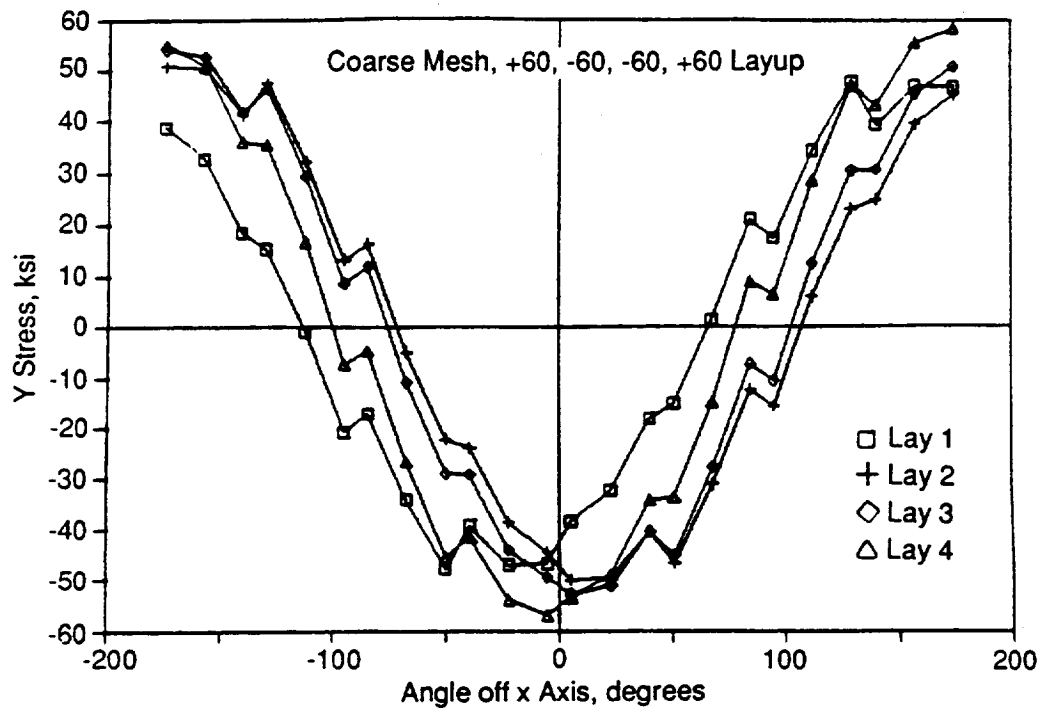


Figure 13. Coarse Duct Axial Stress.

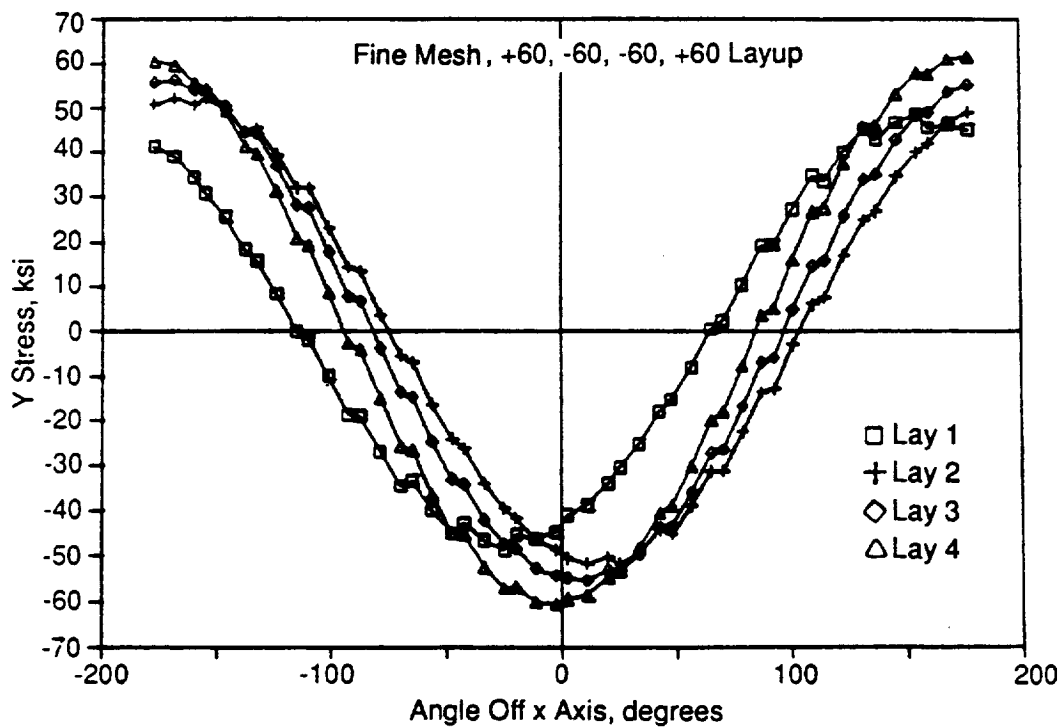


Figure 14. Fine Duct Axial Stress.

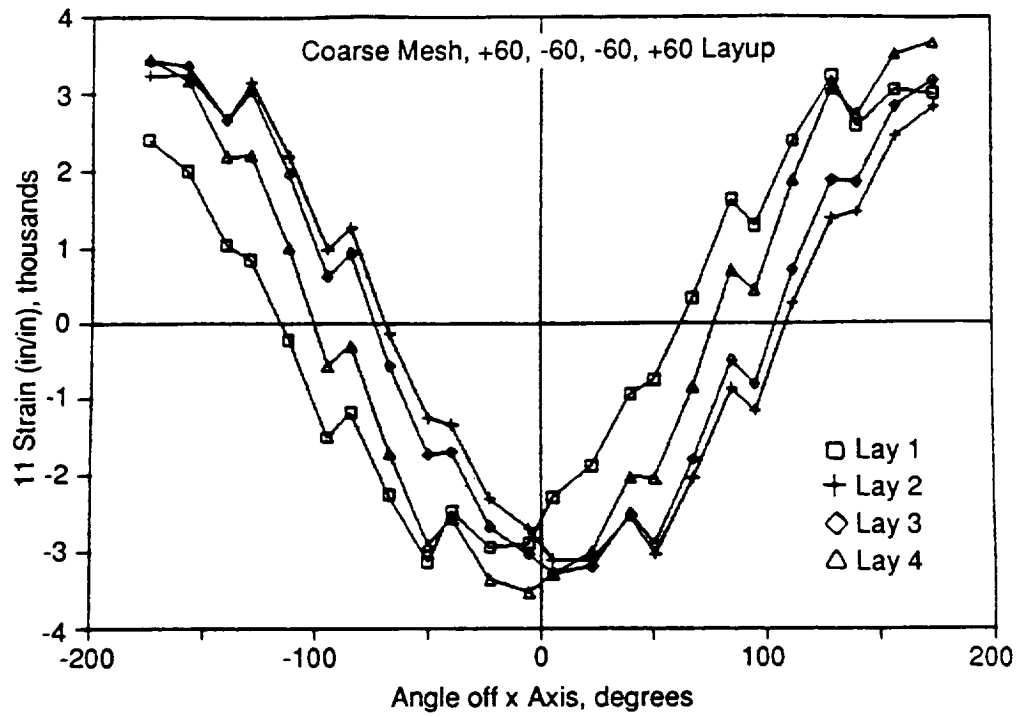


Figure 15. Coarse Duct Material 1 Strain.

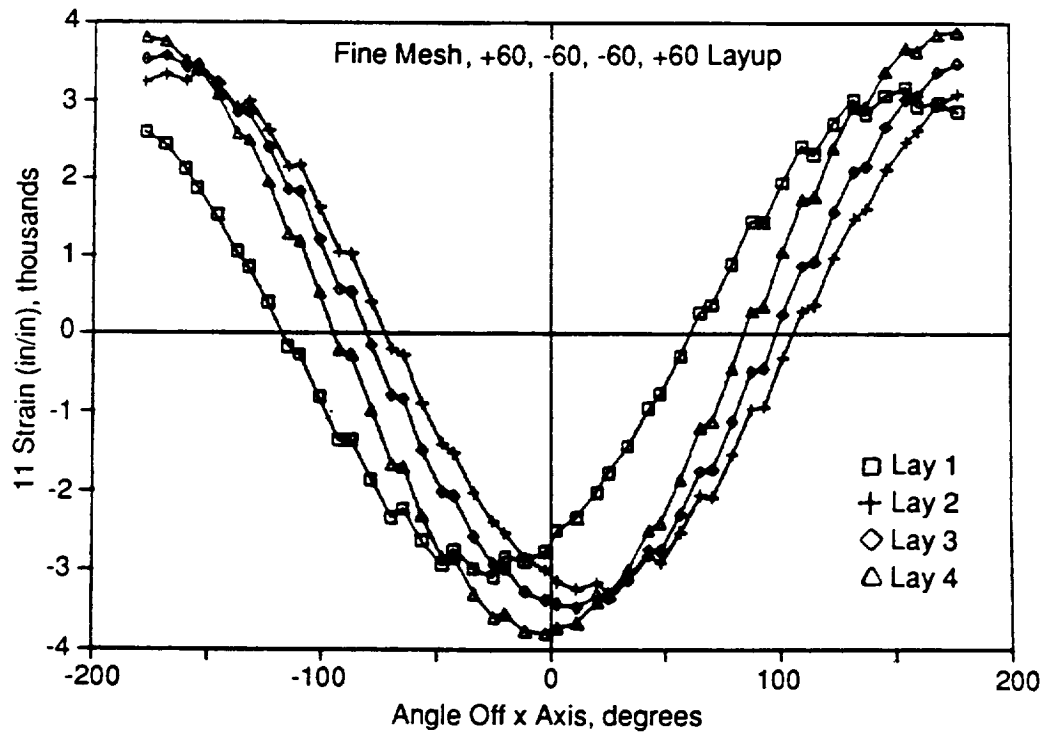


Figure 16. Fine Duct Material 1 Strain.

appears to be reasonable, especially since the fibers are oriented so that they are only 30 degrees off of the duct axis.

2.2.2.2 Fan Frame

A simulated fan frame model comprised of 992 nodes and 88 twenty node bricks was run with overall loads as shown in Figure 17. This simulated frame has an outer case I.D. of 100 inches with an outer case thickness of 3 inches, and a hub ID of 20 inches with a hub thickness of 1 inch. There are 8 equally spaced struts of 1 inch constant thickness.

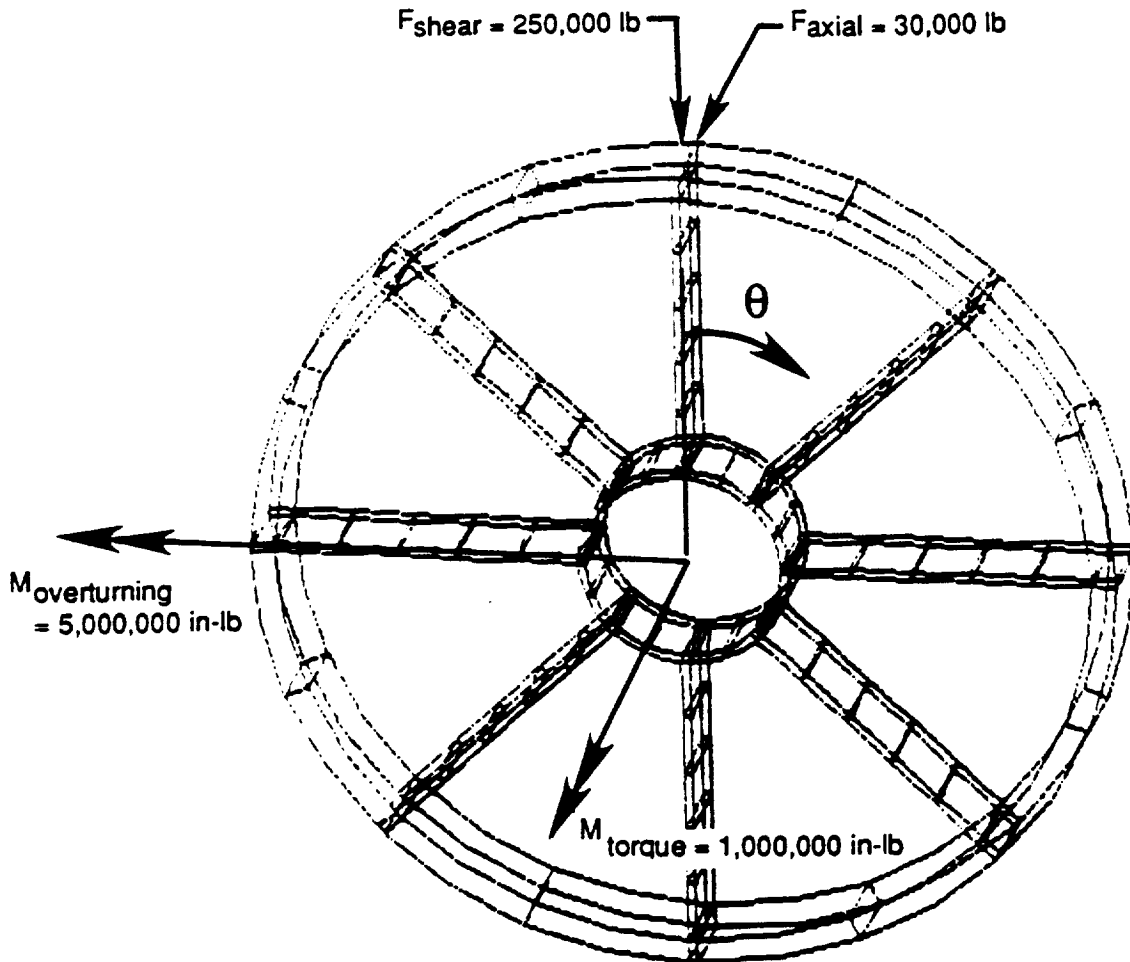


Figure 17. Fan Frame Loads.

The loads were distributed about the outer case rear face with the hub rear face held fixed. The axial force and torque moment were distributed evenly about the outer case while the shear force was distributed about the outer case as a function of $\sin \theta$ and the overturning moment as a function of $\cos \theta$. These loads are typical blade out loads that a frame would have to withstand.

This model was run in CSTEM using isotropic material properties with 3rd order stiffness integration and also with orthotropic layered composite properties. The resulting deflected shape for the isotropic material frame is shown in the two views of Figure 18 with a 10X scale. Figure 19 shows the composite material frame deflections with a 1X scale. The composite frame took about 3 times longer to run than the isotropic frame. The major difference in computation time between the composite and isotropic frame models is in the element stiffness and printout phases, which

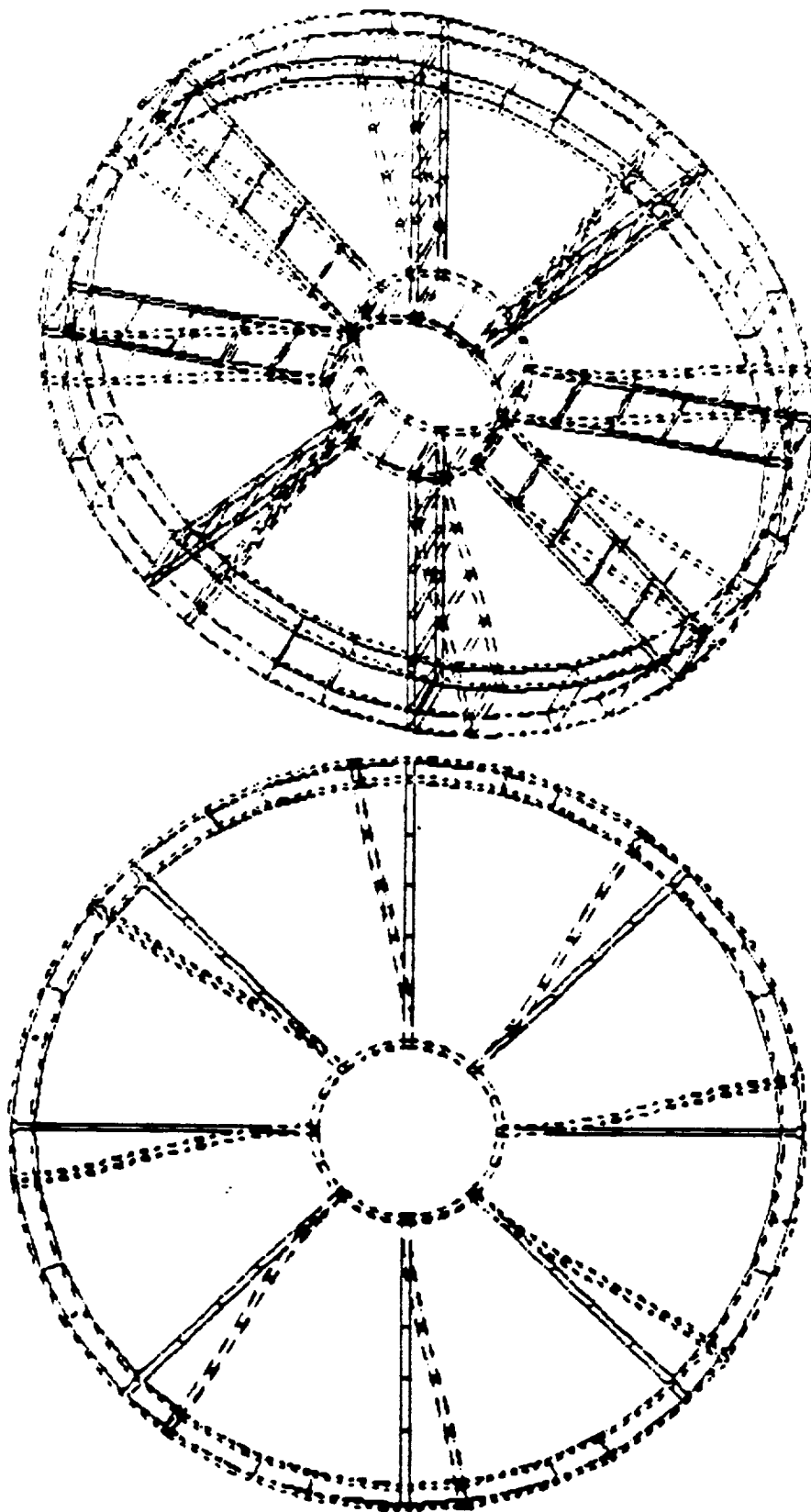


Figure 18. Isotropic Fan Frame Deflected Shape.

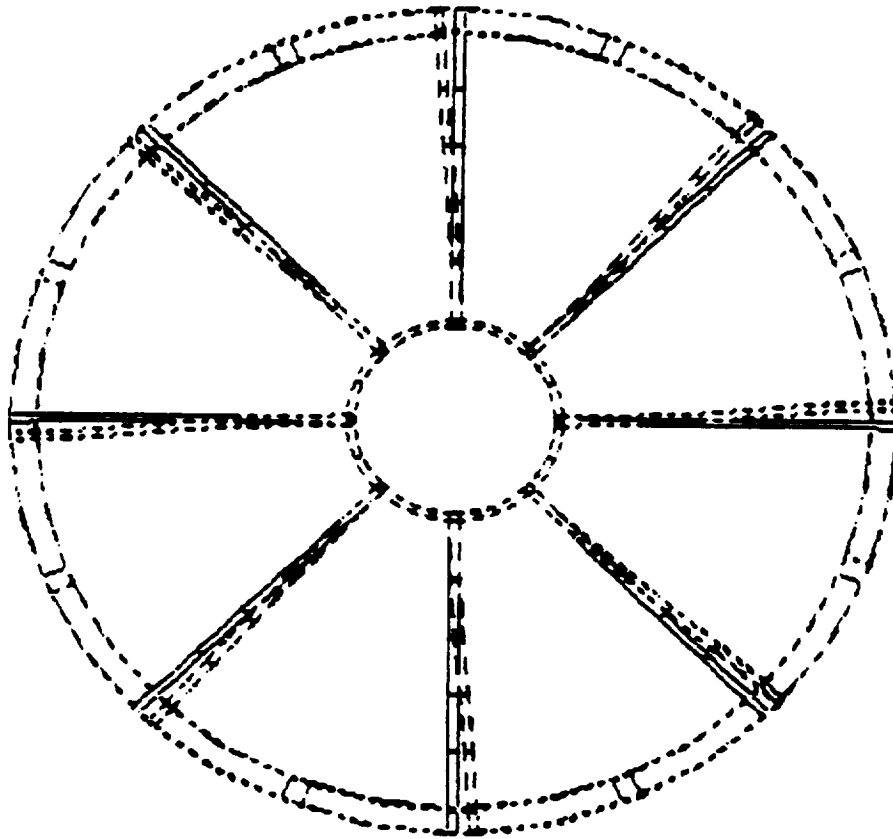


Figure 19. Deflected Composite Fan Frame.

require computations at each integration point. The isotropic frame has 2376 integration points and the composite model has 10512 integration points.

The fan frame with composite properties had a maximum of 22 layers in the outer case elements. The element layering was applied making use of the similarity in the different elements. A representative layup for each of the different elements (outer case, hub, strut) was calculated and then applied to all similar elements. Figure 20 shows the representative layups. Material 2 is a simulated composite material and material 1 is an isotropic material used as a stiff core. The material properties are shown in Figure 21.

2.2.2.3 Fan Blade

A composite fan blade model of an actual unducted fan blade was used as a simulated component. The original model used 8 noded bricks, three through the thickness. This was converted to 20 noded bricks, one through the thickness as shown in Figure 22. The 20 node brick model contains 160 elements and 1263 nodes.

The blade contains three different materials: two composite materials and a titanium core. One composite material is used in 6 outer layers of 0.016 ply thickness. The second composite material is used in 0.030 ply thicknesses internal to the blade. The titanium core extends from the root to about halfway to the tip for about half of the chord at the center of the blade and is tapered so that it is covered by roughly 0.21 inches of composite material. The composite material layers were input as the actual thicknesses, while the titanium material was included in 0.030 thicknesses. This

Number of Layups to be Formed: 3

Layup Number 1

Beginning ELEM No. 20 to 0 C/S Axis: 3 Gen Order Used: 3

Resulting 1/2 Layup No. 1

Cross Section Begins with Element 20 Along T Axis

Total Thickness = 3.0015

Layer	Matl.	Thick. Fraction	Angle	LTRAN
1	2	0.33317E-01	30.000	0
2	2	0.33317E-01	-30.000	0
3	2	0.33317E-01	30.000	0
4	2	0.33317E-01	-30.000	0
5	2	0.33317E-01	30.000	0
6	2	0.33317E-01	-30.000	0
7	2	0.33317E-01	30.000	0
8	2	0.33317E-01	-30.000	0
9	2	0.33317E-01	30.000	0
10	2	0.33317E-01	-30.000	0
11	1	0.16658	0.00000	0
12	1	0.16658	0.00000	0
13	2	0.33317E-01	-30.000	0
14	2	0.33317E-01	30.000	0
15	2	0.33317E-01	-30.000	0
16	2	0.33317E-01	30.000	0
17	2	0.33317E-01	-30.000	0
18	2	0.33317E-01	30.000	0
19	2	0.33317E-01	-30.000	0
20	2	0.33317E-01	30.000	0
21	2	0.33317E-01	-30.000	0
22	2	0.33317E-01	30.000	0

Layup Number 2

Beginning ELEM No. 4 to 0 C/S Axis: 3 Gen Order Used: 2

Resulting 1/2 Layup No. 2

Cross Section Begins with Element 4 Along T Axis

Total Thickness = 0.99944

Layer	Matl.	Thick. Fraction	Angle	LTRAN
1	2	0.10006	0.00000	0
2	2	0.10006	90.000	0
3	2	0.10006	0.00000	0
4	2	0.10006	90.000	0
5	1	0.99777E-01	0.00000	0
6	1	0.99777E-01	0.00000	0
7	2	0.10006	90.000	0
8	2	0.10006	0.00000	0
9	2	0.10006	90.000	0
10	2	0.10006	0.00000	0

Layup Number 3

Beginning ELEM No. 45 to 0 C/S Axis: 3 Gen Order Used: 1

Resulting 1/2 Layup No. 3

Cross Section Begins with Element 45 Along T Axis

Total Thickness = 1.0000

Layer	Matl.	Thick. Fraction	Angle	LTRAN
1	2	0.10000	60.000	0
2	2	0.10000	-60.000	0
3	2	0.10000	60.000	0
4	2	0.10000	-60.000	0
5	1	0.10000	0.00000	0
6	1	0.10000	0.00000	0
7	2	0.10000	-60.000	0
8	2	0.10000	60.000	0
9	2	0.10000	-60.000	0
10	2	0.10000	60.000	0

Figure 20. Representative Layups of Outer Case, Hub, and Strut.

Material Elastic Properties													
MTN	RTEM	MTP	DEN										
1	70.0	1	0.2980										
Temperature	EXX	EYY	EZZ	PXY	PYZ	PXZ	GYX	GYZ	GXZ	ALX	ALY	ALZ	
70.0	30.0	30.0	30.0	0.31	0.31	0.31	11.4	11.4	11.4	0.0	0.0	0.0	
MTN	RTEM	MTP	DEN										
2	70.0	1	0.0554										
Temperature	EXX	EYY	EZZ	PXY	PYZ	PXZ	GYX	GYZ	GXZ	ALX	ALY	ALZ	
70.0	18.8	1.2	1.2	0.26	0.42	0.26	0.6	0.4	0.5	-0.2	15.3	15.3	

Figure 21. Composite Frame Material Properties.

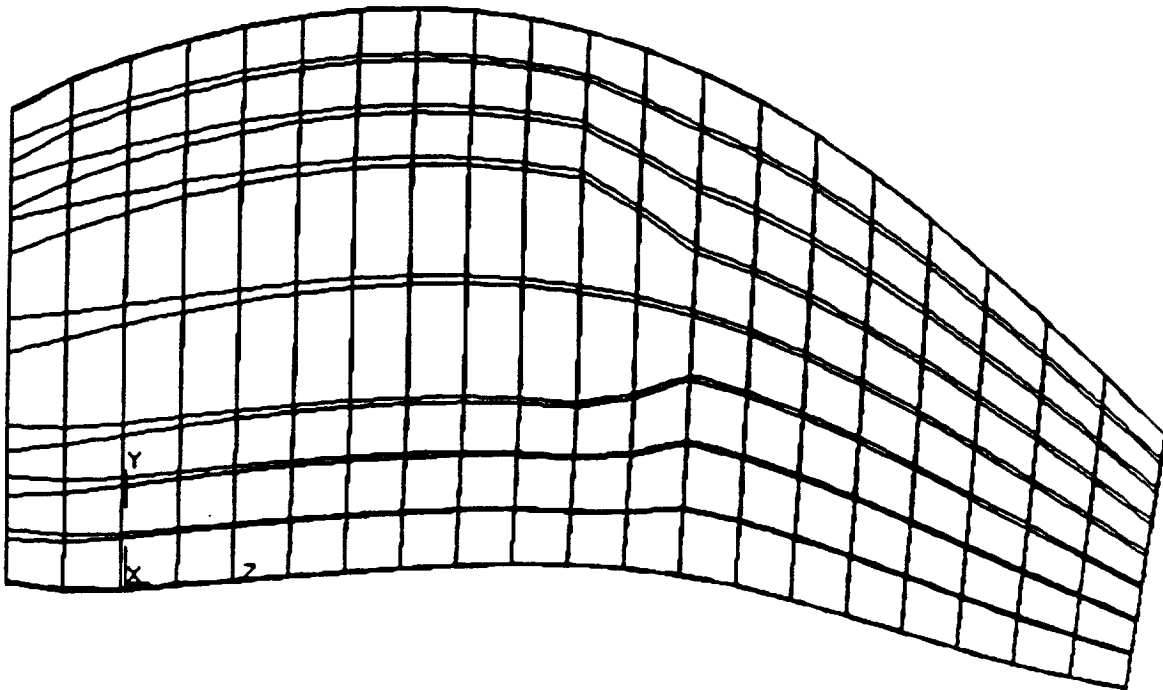


Figure 22. Composite Fan Blade.

resulted in the number of layers per element ranging from 6 at the tip trailing edge to 76 at the root midchord.

Free vibration frequencies of a test blade are available to compare with this model. The first 8 free vibration modes were calculated in CSTEM and are shown in Figures 23 through 30 along with the test results. The CSTEM results were obtained using 370 CPU seconds on the CRAY-XMP. It can be seen that the agreement with the test results is very good. These experimental test blade results can be scaled to the full scale blade. Table 3 contains the scaled full size blade experimental frequencies.

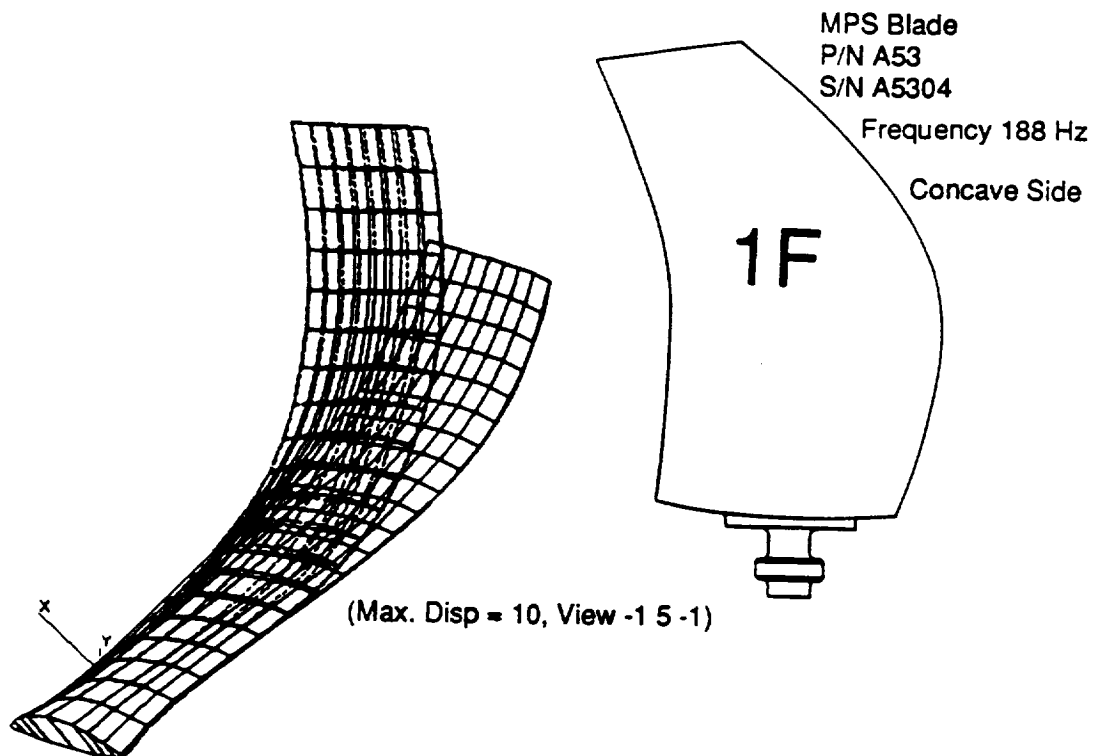


Figure 23. Mode 1, Predicted Frequency = 173.48 CPS.

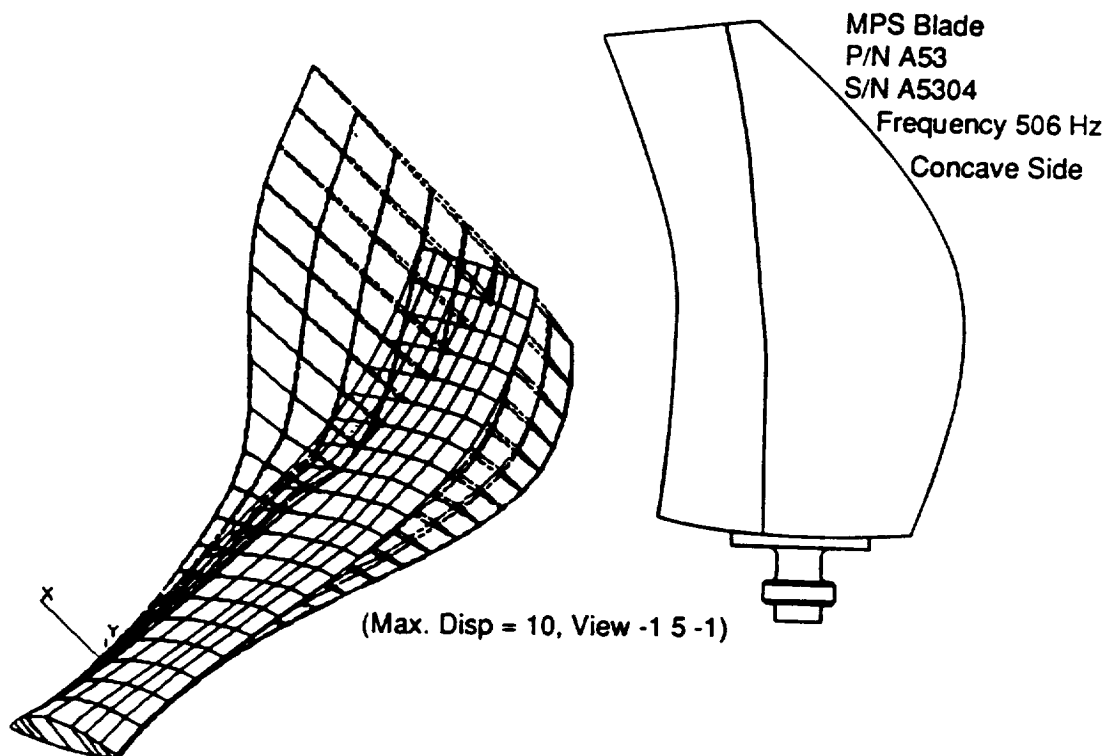


Figure 24. Mode 2, Predicted Frequency = 506.72 CPS.

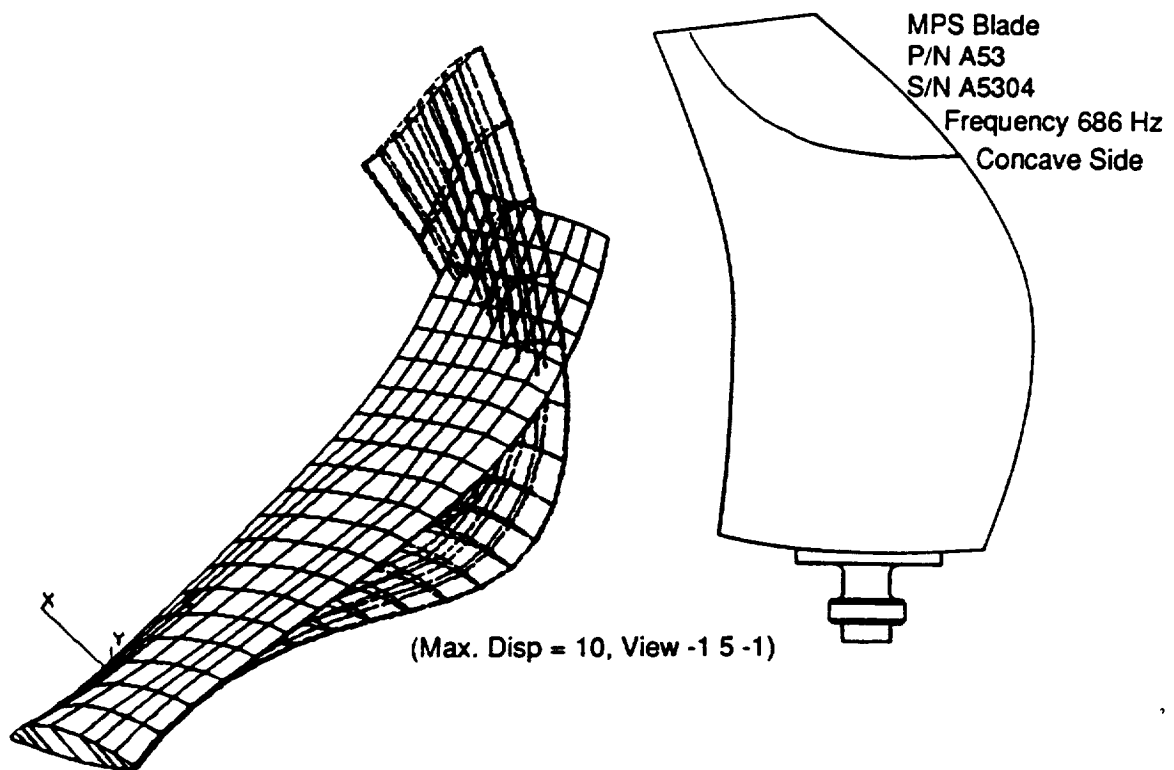


Figure 25. Mode 3, Predicted Frequency = 618.66 CPS.

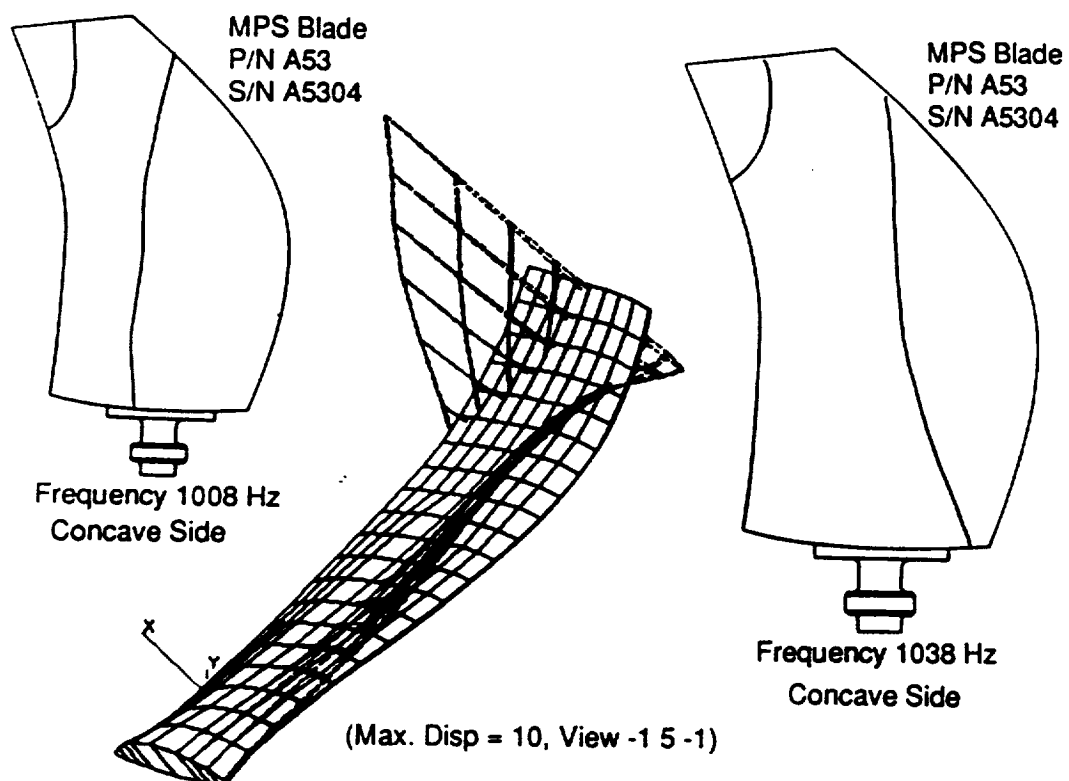


Figure 26. Mode 4, Predicted Frequency = 1083.60 CPS.

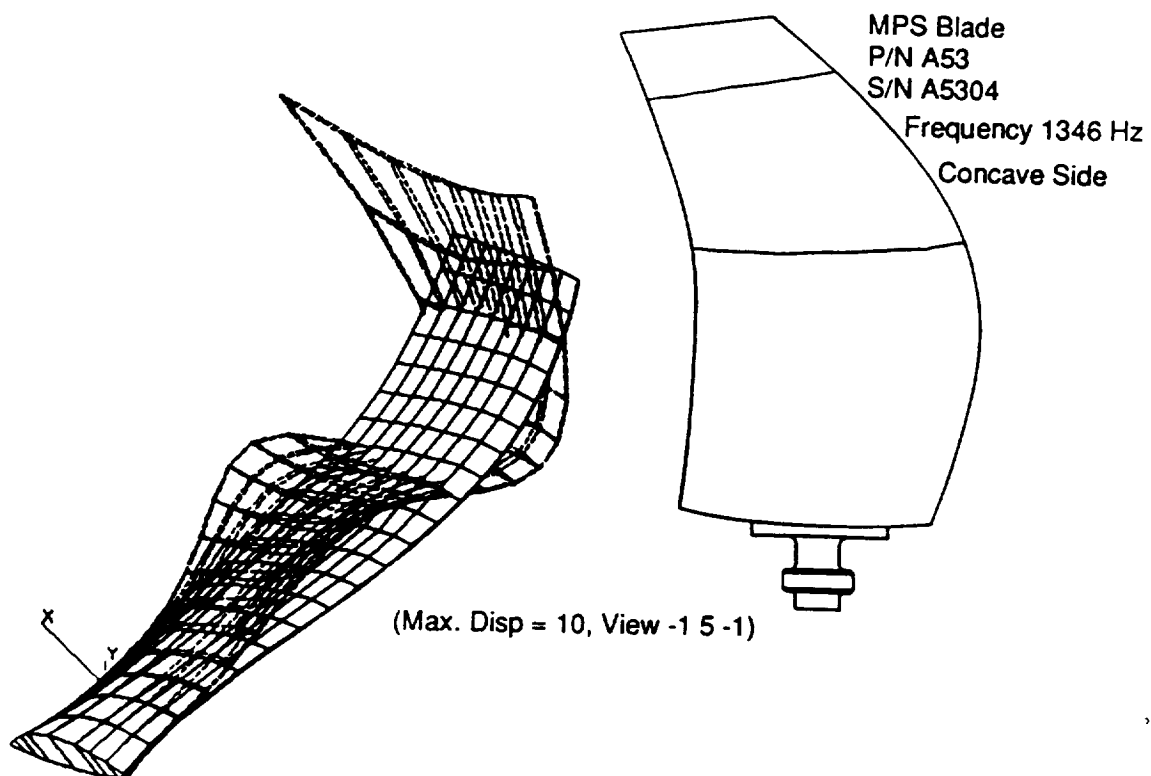


Figure 27. Mode 5, Predicted Frequency = 1229.57 CPS.

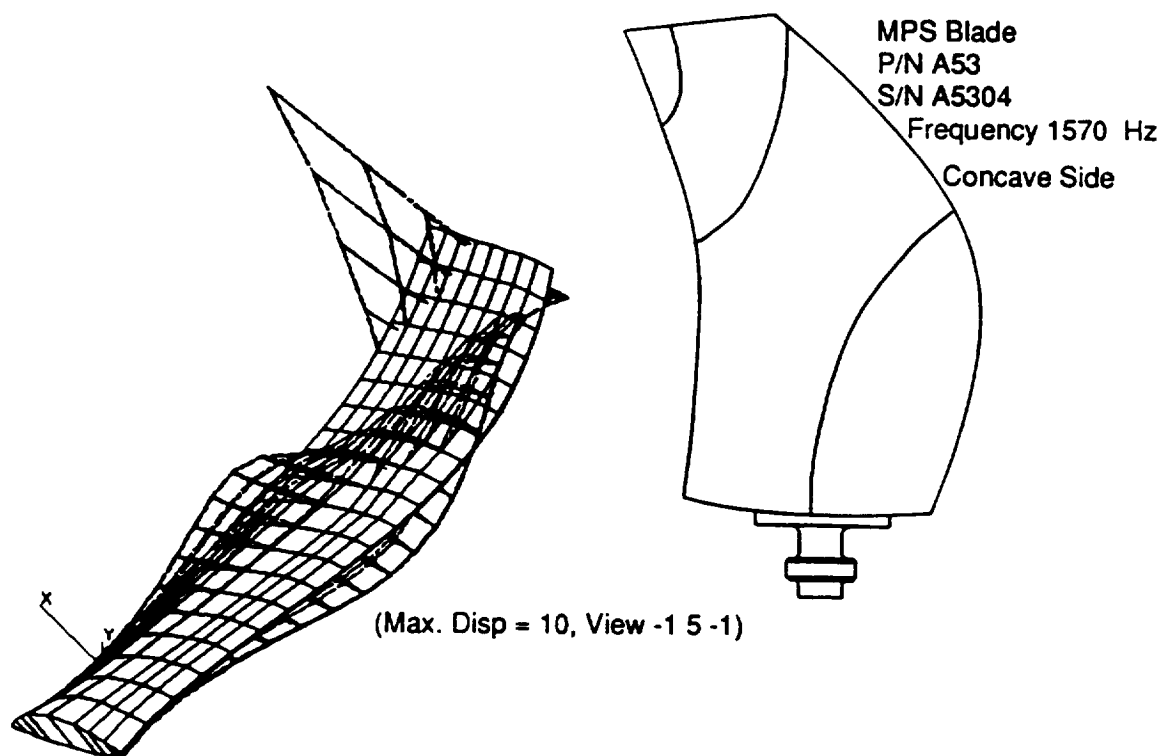


Figure 28. Mode 6, Predicted Frequency = 1632.29 CPS.

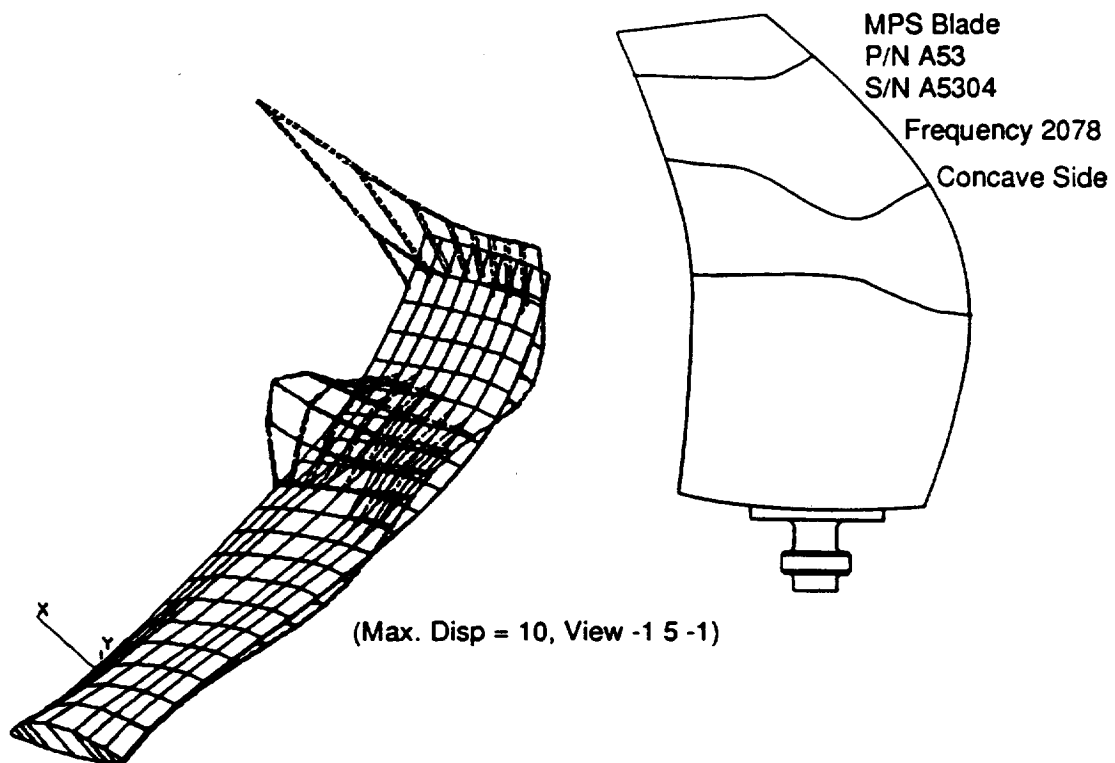


Figure 29. Mode 7, Predicted Frequency = 1940.58 CPS.

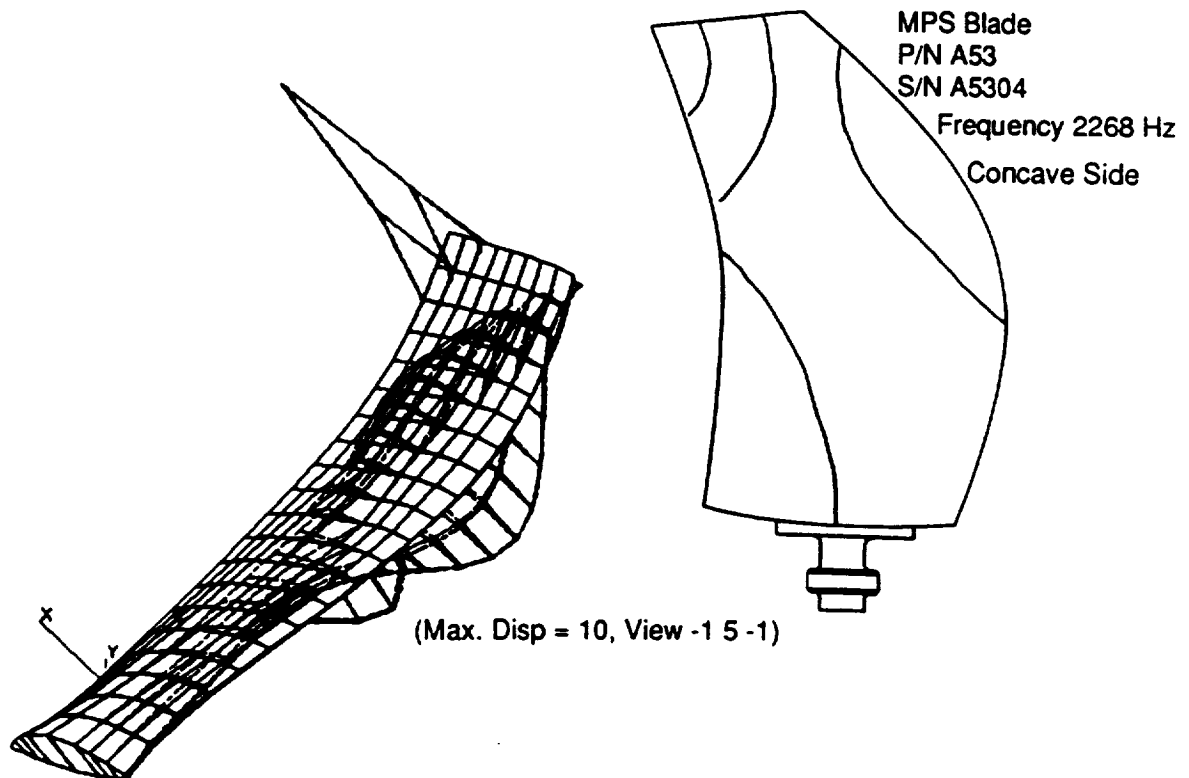


Figure 30. Mode 8, Predicted Frequency = 2309.59 CPS.

Table 3. Composite Fan Blade Frequencies.

Mode	Linear	CSTEM		Previous Analysis		Test
		4 L.C.	eq iter	Linear	Stiffened	
1	31.05	51.84	51.78	33.1	49.868	33.65
2	90.69	105.11	105.77	95.5	106.650	90.57
3	110.70	137.98	137.40	121.3	150.589	122.79
4	193.92	212.98	214.09	204.1	224.608	185.80
5	219.98	248.71	248.00	245.5	275.572	240.93
6	292.08	315.14	316.91		345.681	281.03
7	347.02	392.74	392.03		444.276	371.96
8	413.34	445.67	447.75		481.521	405.97

The full scale model of the composite unducted fan blade model was run with a centrifugal loading of 1341 RPM, which is the actual speed for this blade. Two different updated Lagrange large displacement analyses were done at this speed as well as a linear analysis.

One of the large displacement analyses used no equilibrium iterations, but stepped up to 1341 RPM in 3 load cases (500, 1000, 1341) with a fourth load case at 1341 RPM so that the full stiffening effects would be included. The other large displacement analysis used a single load case to 1341 RPM, but with 5 equilibrium iterations. This analysis converged to a displacement difference between iterations of 1.78E-3.

The 4 load case large displacement analysis resulted in a max tip deflection of 0.6883 inches and took 2820 CPU seconds on the CRAY. The equilibrium iteration large displacement analysis gave a max tip deflection of 0.7536 inches, which took 2351 CPU seconds on the CRAY. The linear analysis gave a max tip deflection of 1.423 inches and took 508 CPU seconds. The calculated frequencies for each of these cases is shown in Table 3. As a comparison, the frequencies calculated in a previous GE analysis of this blade using a different code are also listed in Table 3.

Also available for comparison are mode shape slopes, which are used in aeromechanical stability analyses of the blade. Figures 31 through 36 show a comparison of calculated mode shape slopes at 83% span between the previous GE analysis and the CSTEM analysis for the first 6 stiffened vibration modes. Agreement between the results is very good. A less tangible, but no less significant comparison might be made in that the previous results require a great deal more manipulation of data requiring several different computer programs to be run, while the CSTEM code is essentially self contained.

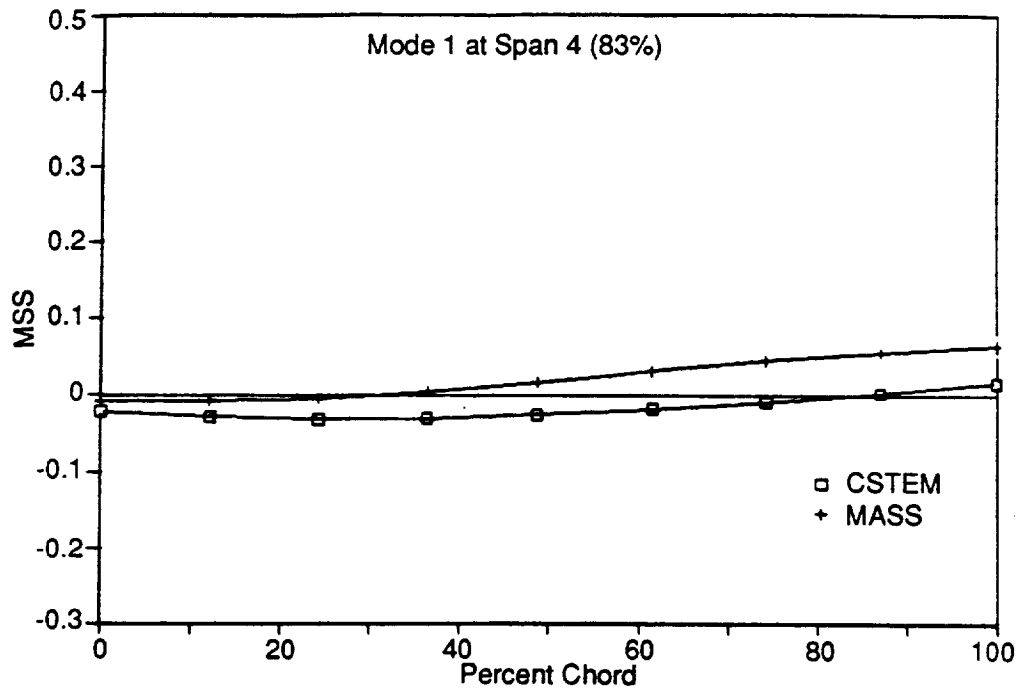


Figure 31. Mode Shape Slope at 1341 RPM, Mode 1.

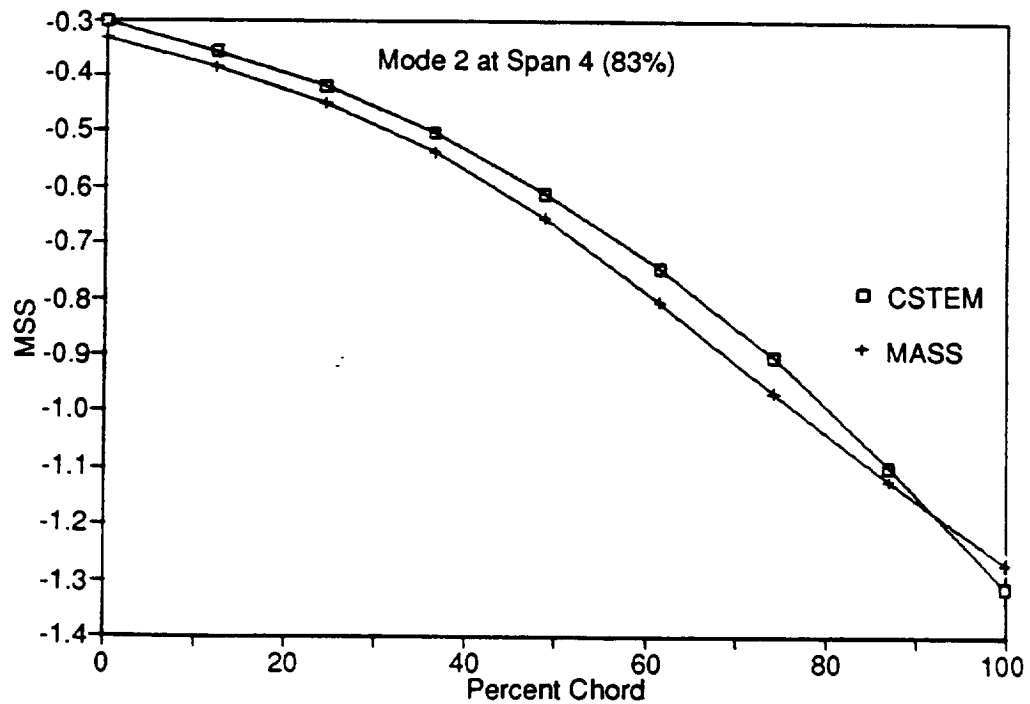


Figure 32. Mode Shape Slope at 1341 RPM, Mode 2.

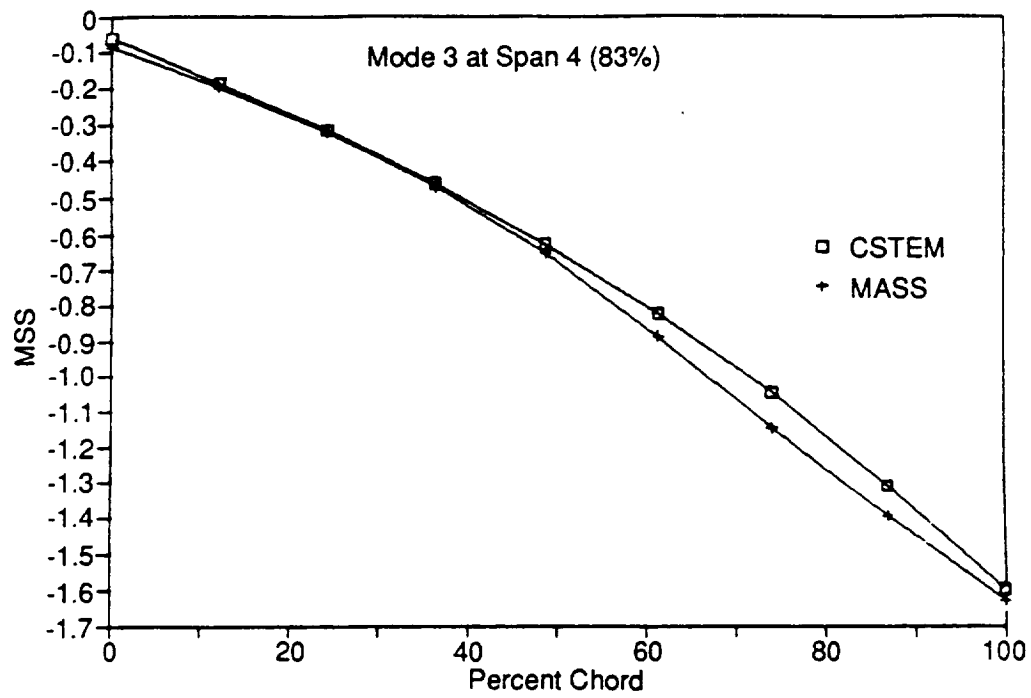


Figure 33. Mode Shape Slope at 1341 RPM, Mode 3.

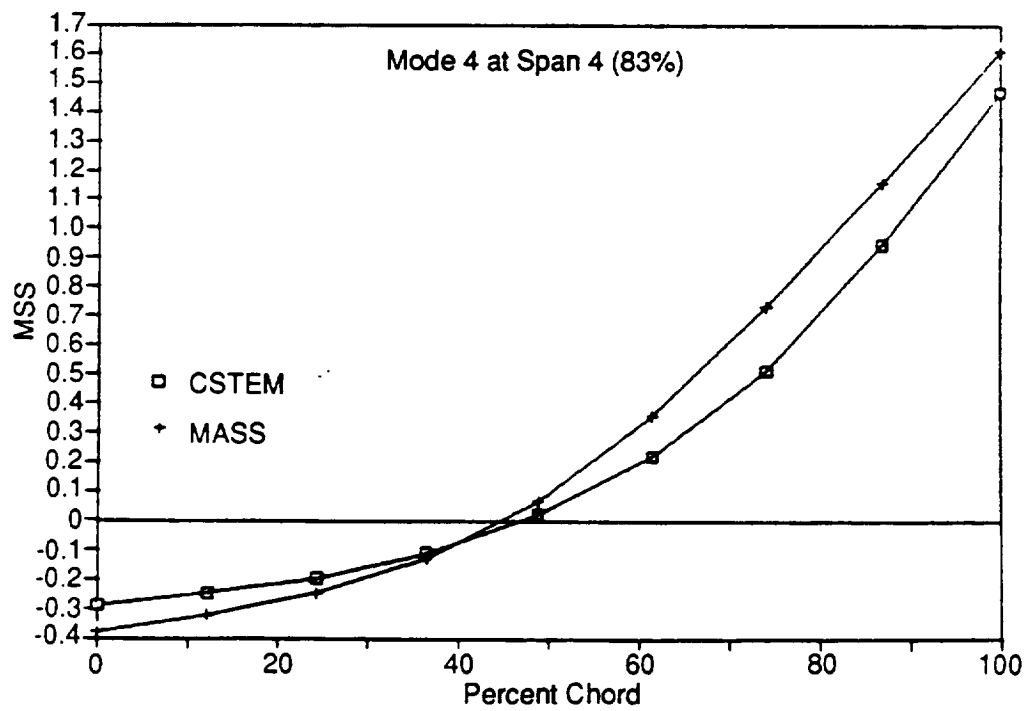


Figure 34. Mode Shape Slope at 1341 RPM, Mode 4.

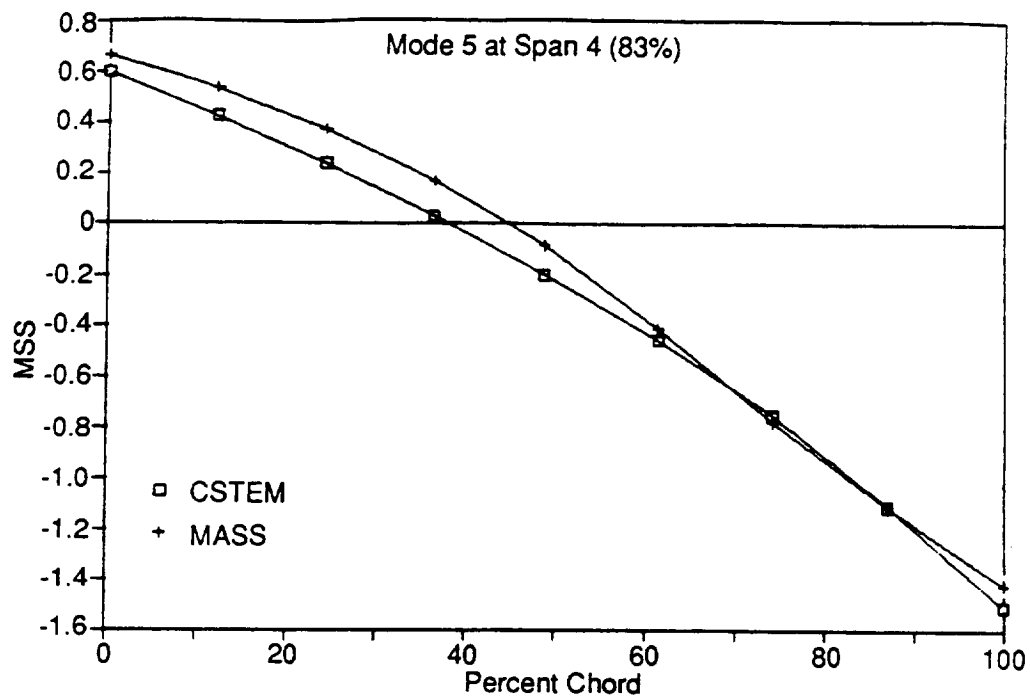


Figure 35. Mode Shape Slope at 1341 RPM, Mode 5.

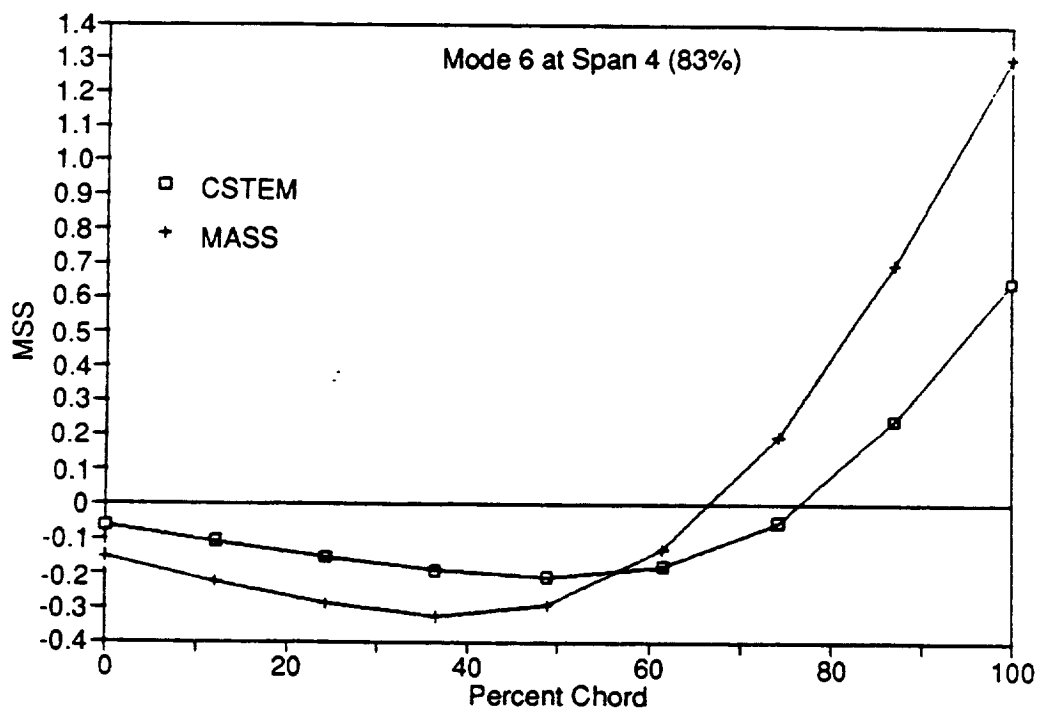


Figure 36. Mode Shape Slope at 1341 RPM, Mode 6.

2.2.2.4 Turbine Frame

A simulated turbine rear frame was analyzed by CSTEM. This model consists of 1288 nodes and 147 elements. The hub has an I.D. of 2 feet and a thickness of 1 inch. The outer case has an I.D. of 5 feet and a thickness of 6 inches. There are 7 struts of 1 inch thickness that attach to the hub at a 45 degree angle. There is also a 1 inch thick cone that is attached to the hub forward face which reduces from a 2 foot I.D. to a 1 foot I.D. to simulate a bearing support. The model is pictured in Figure 37.

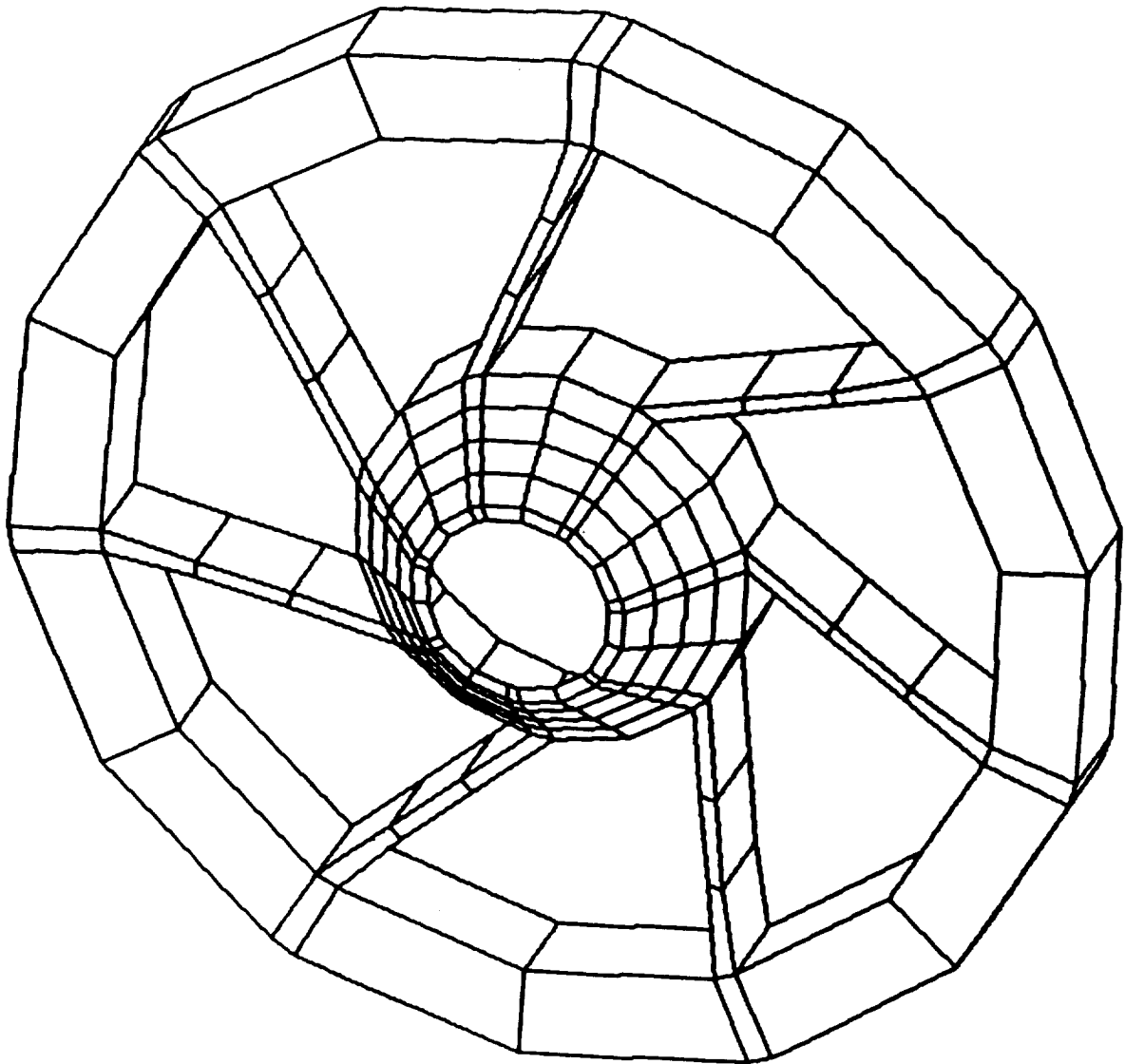


Figure 37. Simulated Turbine Rear Frame.

Structural loadings were typical aft mount loads of 7000 lbs. applied to the outer case at 26 degrees on either side of the top. The outer case O.D. was fixed from moving outward as was the forward most I.D. of the support cone. Structural solutions were performed at the 12 and 30 second timepoints. Figures 38, 39, and 40 show results from this analysis, which took 390 CPU seconds on the CRAY-XMP.

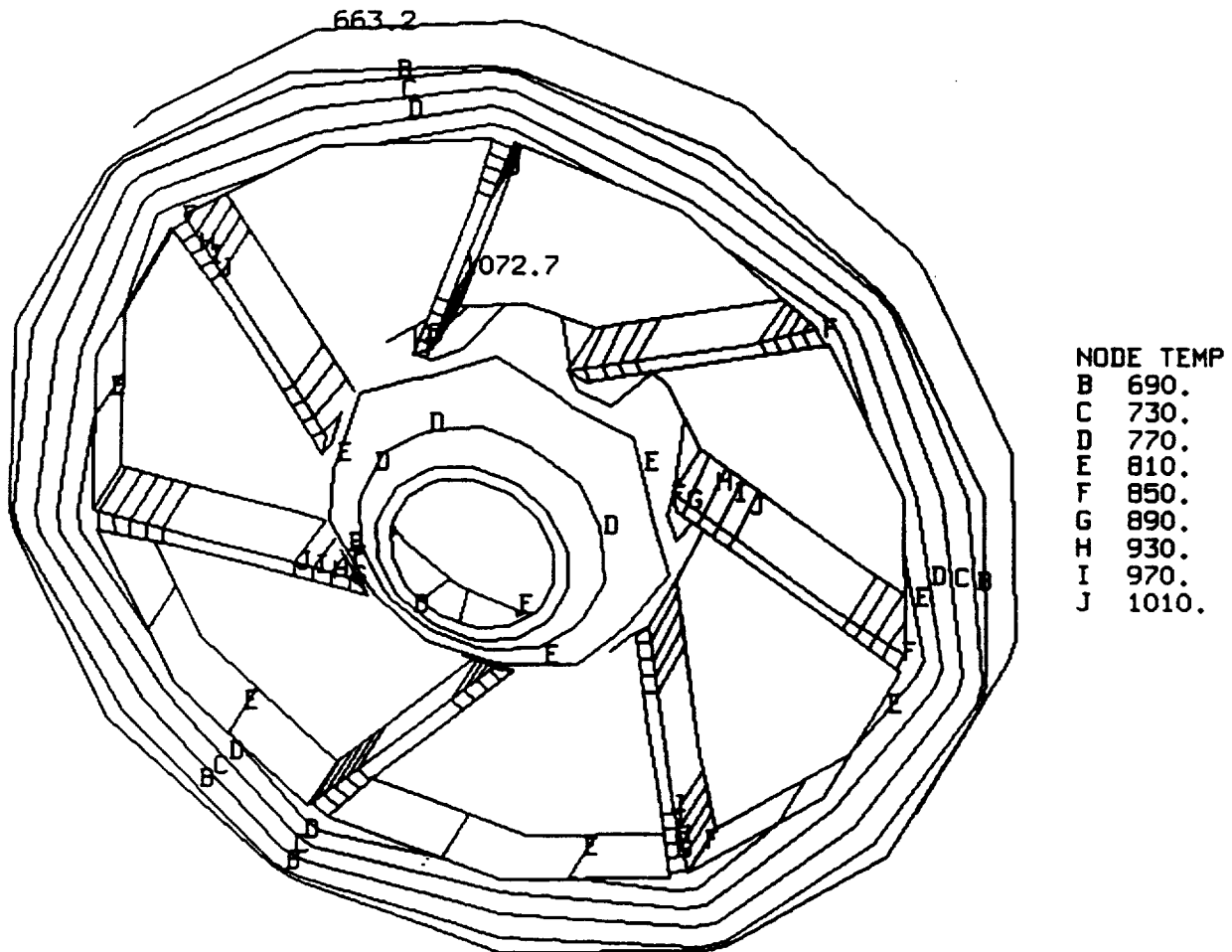


Figure 38. Temperature Distribution at 12 Seconds.

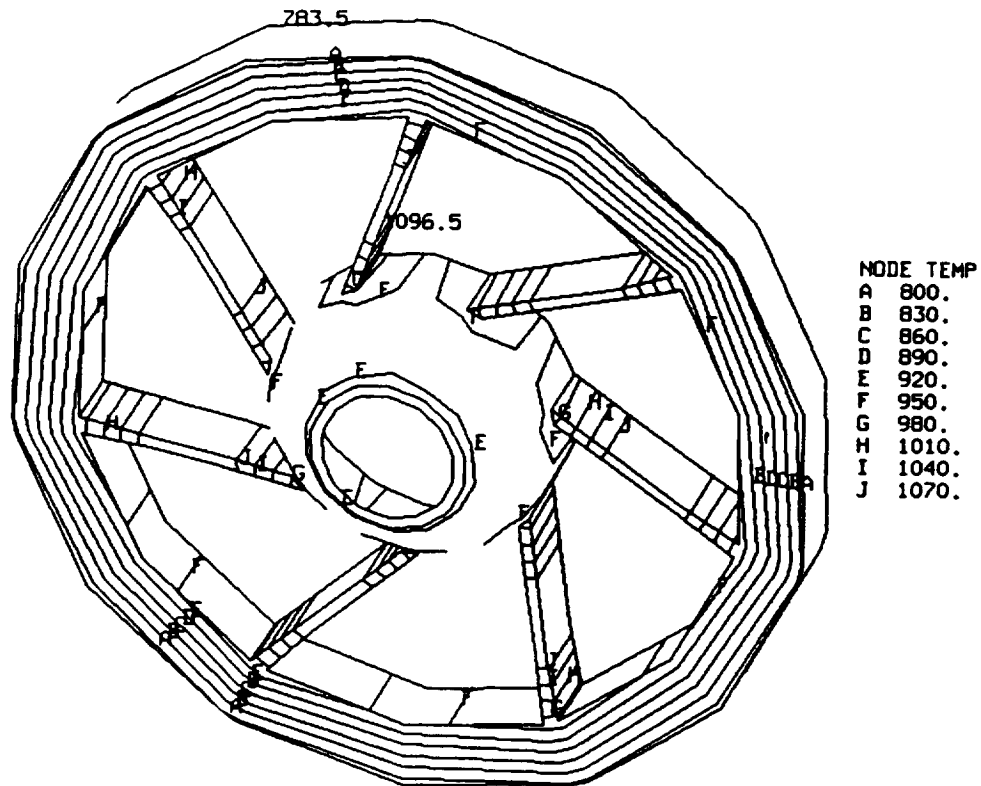


Figure 39. Temperature Distribution at 30 Seconds.

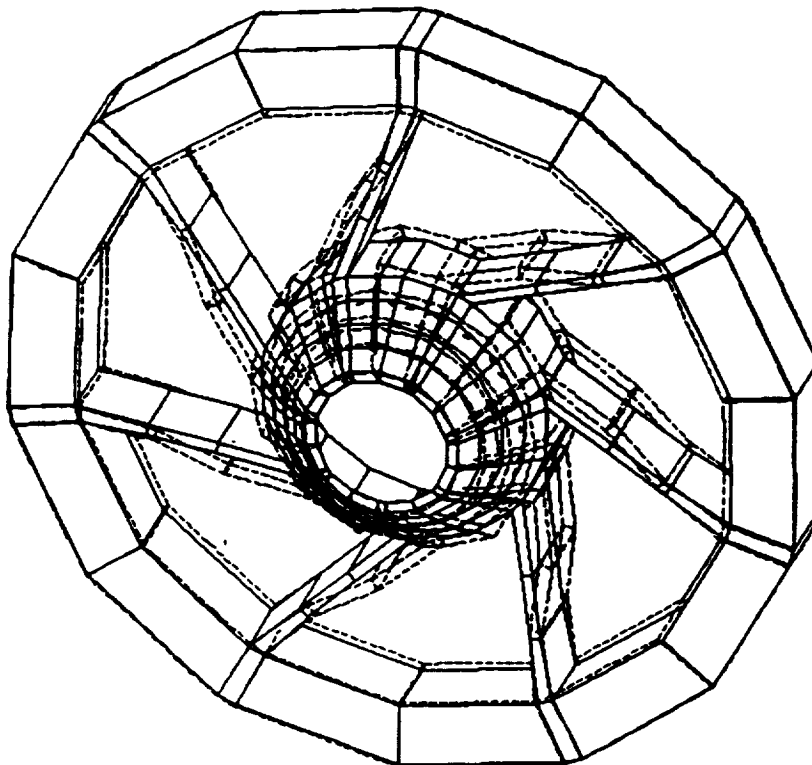


Figure 40. Deflected Shape (10X) at 30 Seconds.

2.2.2.5 Turbine Blade

A heat transfer analysis of an actual turbine blade was performed and compared with results from a heat transfer code currently used at GE. The blade is a hollow blade and must be modeled in slices for use in the other heat transfer code. This restriction is not necessary in CSTEM, but for comparison purposes the same sliced model was used. The blade slice model is shown in the Figure 41 inset. The model consists of 924 nodes and 338 elements. The elements used are 8 node bricks. The material is René N5, an isotropic material.

A nonlinear transient analysis of the blade slice was performed with results as shown in the contour plot of Figure 42. The comparison between these results and those obtained previously is shown in Figure 41. As can be seen the results are nearly identical with those obtained previously from the other code.

A simulated turbine blade is also being examined so that a more complete analysis may be demonstrated. The model to be used is shown in Figure 43. This model consists of 503 nodes and 60 elements, which are 20 node bricks. This is basically a scaled up model of an existing SSME blade model that was available. The overall span of the simulated blade is roughly 2 inches with a tip radius of approximately 13.8 inches.

2.3 Tailoring

The tailoring function of CSTEM can handle a number of different procedures with the same general program. This is accomplished by a software interface between the tailoring function and the structural, acoustical, heat transfer routines, etc. This interface is a FORTRAN subroutine ANALIZ. There is an input file read by COPEs to specify the variables used and type of tailoring to perform. The ANALIZ routine rereads this input file and handles the input and output between the analysis modules involved and the tailoring logic. This is done by calling specific routines depending on the specified tailoring procedure. The currently available tailoring procedures are described in the following example test cases and simulated component cases.

2.3.1 Test Cases

2.3.1.1 Absorption Tailoring

The object function is electromagnetic absorption. This function is to be maximized for a finite element model constructed of layers of different composite materials. The thickness of each of the layers are used as design variables. These thicknesses are expressed in terms of the element thickness fractions. Figure 44 shows a simple cantilever beam test case consisting of 3 initially equal thickness layers of different material with a 10 GHz normally incident electromagnetic wave. The energy absorbed increased from 6.2% to 11.1% while the layer thickness fractions changed from 0.333 for each layer to 0.09, 0.54, and 0.37. Figure 45 shows the percentage change in absorption for each evaluation throughout the tailoring process. The total analysis took 3 CPU seconds on the CRAY-XMP.

2.3.1.2 Acoustic Tailoring

The object function is noise generated by a finite element model subject to sound source of a specific magnitude and frequency. In the case considered the structure is composed of 4 equal thickness layers of a composite material subject to a 565 cps source at 4000 dB. The angles of the layers are the design variables and their variations produce natural frequencies and mode shapes to minimize the reflected sound. The noise was reduced from $7.16\text{E-}6$ watts to $2.13\text{E-}7$ watts. The final layer angles are 79, 90, 90, 79 degrees. Figure 46 shows the model used and Figure 47 shows the

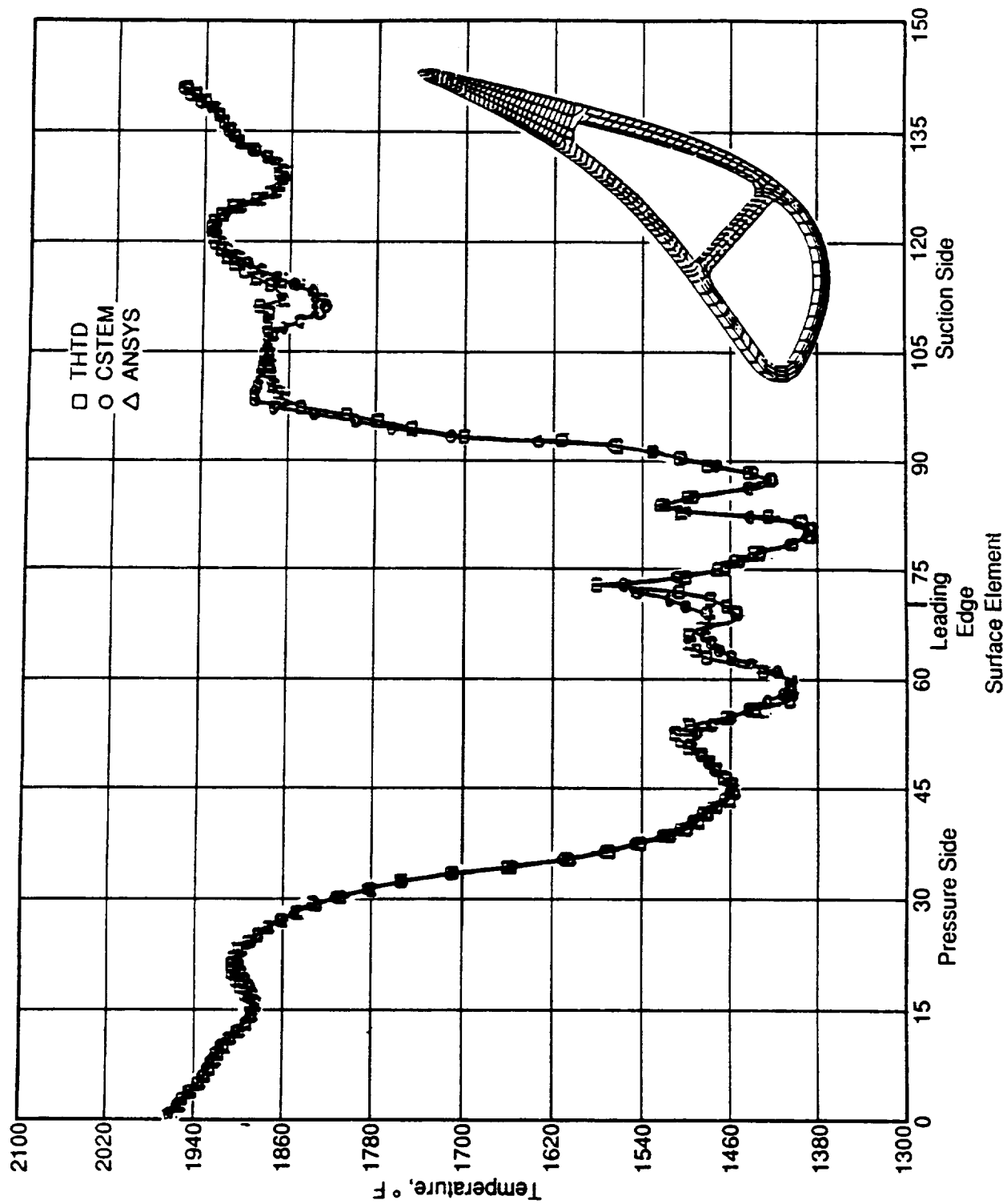


Figure 41. CSTEM Versus THT-D and ANSYS Results.

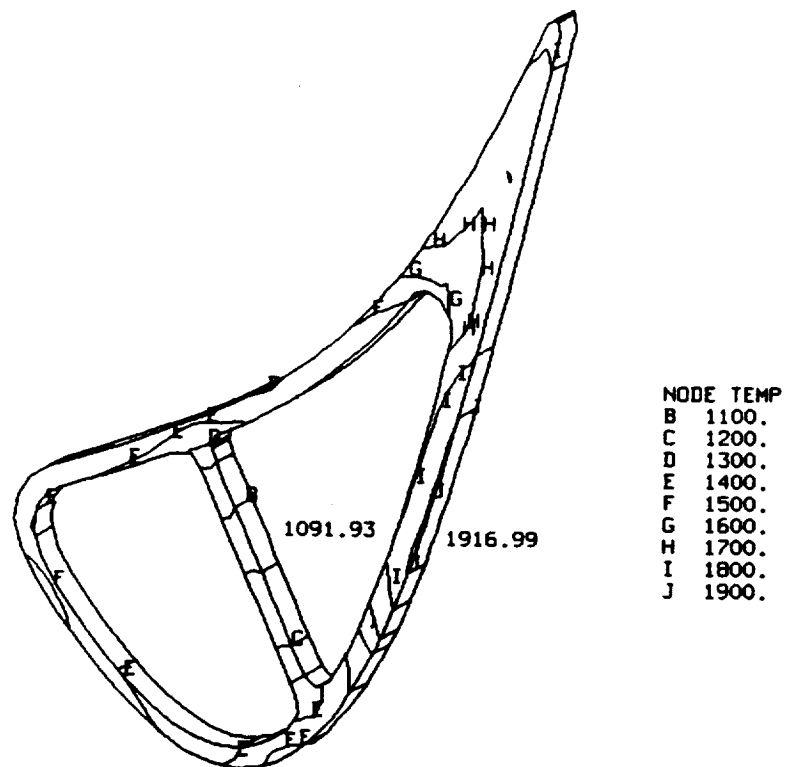


Figure 42. CSTEM Heat Transfer Results for Turbine Blade Slice.

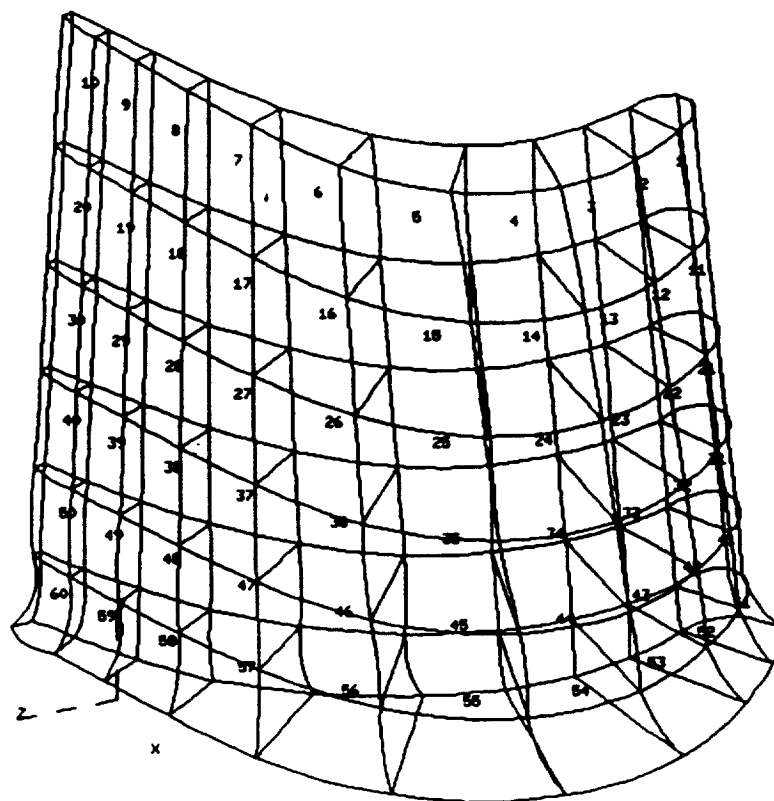


Figure 43. Simulated Turbine Blade (3,2,1).

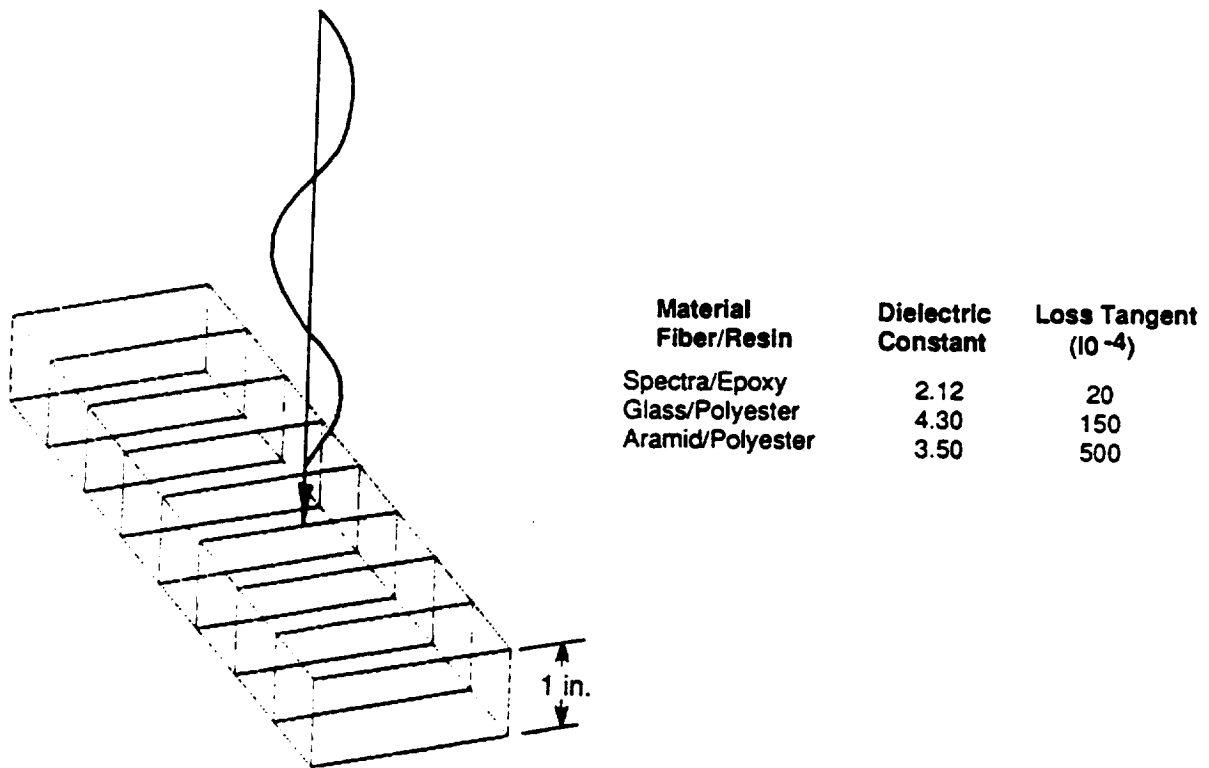


Figure 44. Absorption Tailoring Model.

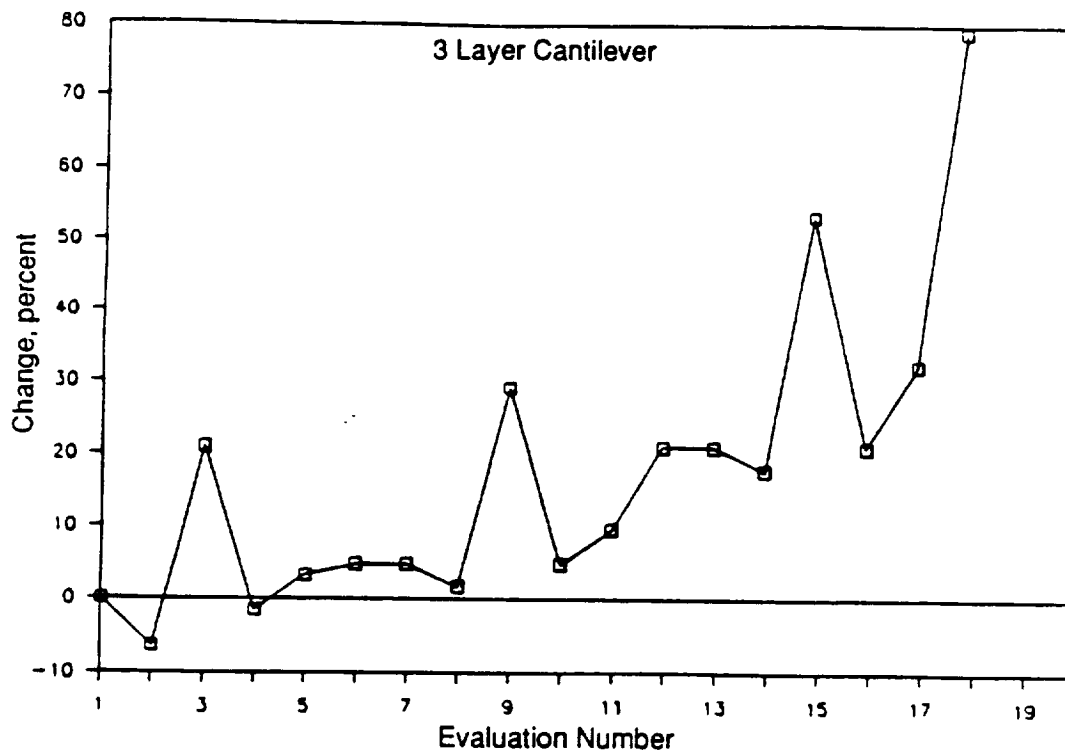


Figure 45. Absorption Tailoring Results.

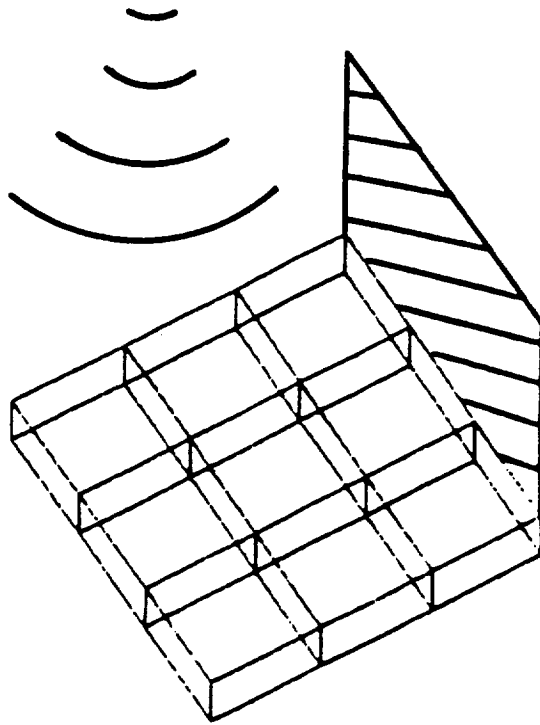


Figure 46. Acoustic Tailoring Model.

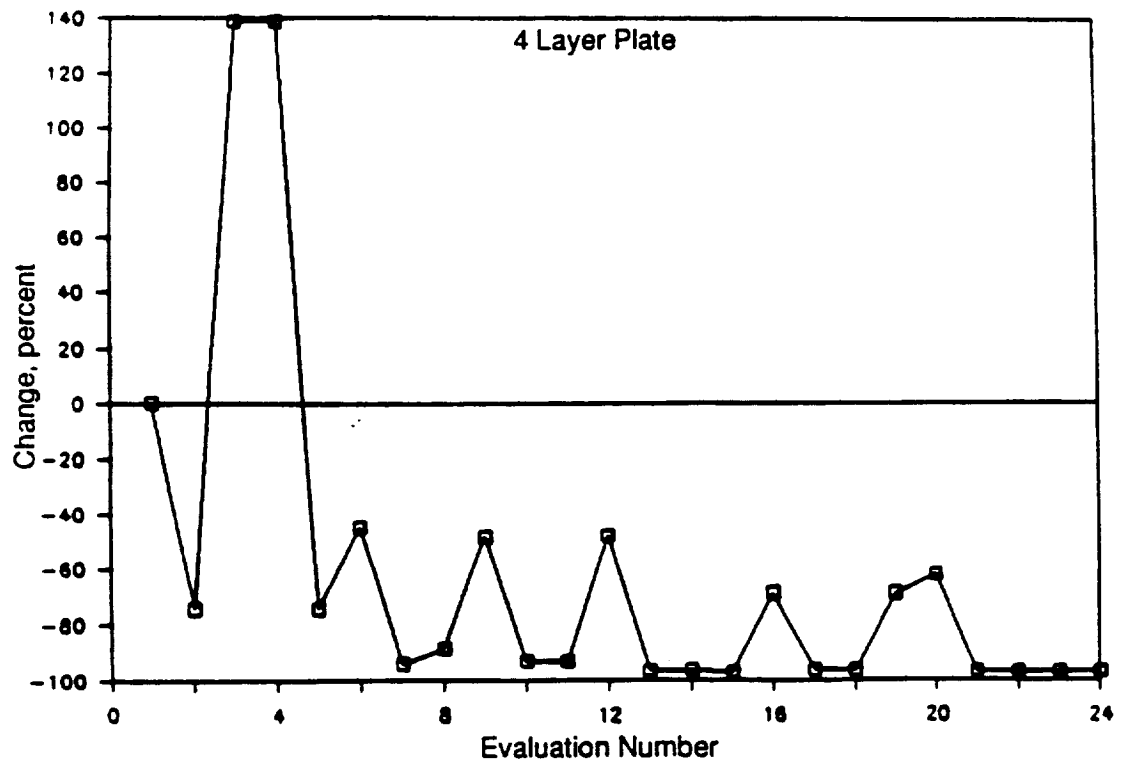


Figure 47. Acoustic Tailoring Results.

percentage change in the noise with evaluation number. This case took 200 CPU seconds on the CRAY-XMP.

2.3.1.3 Frequency Tailoring

The object function is the lowest of the first 3 natural frequencies of a finite element model. As with acoustic tailoring, the model considered is composed of 4 equal thickness layers of a composite material. The angles of these layers are the design variables. They are changed to minimize the object function. The other 2 frequencies are constraints. The fundamental frequency was lowered from 561 cps to 351 cps, with final layer angles of 25, 51, 51, 25 degrees. Figure 48 shows the model used and Figure 49 shows the percentage change in the frequency with evaluation number. This took 71 CPU seconds on the CRAY-XMP

2.3.1.4 Weight Tailoring

The object function is the weight of the model. For the weight tailoring the finite element model is a cantilever beam 8 inches long and 3 inches thick. The design variables are the layer thicknesses (expressed as element thickness fractions), layer orientation, and fiber volume ratios. Use of the fiber volume ratio as a design variable is done by calculating ply material properties with the INHYD routines. One constraint is used: the lowest natural frequency is to be larger than 200 cps. Initial thickness fractions of 0.30, 0.25, 0.45; angles of 90, 90, 45 degrees; fiber volume ratios of 0.6, 0.6, 0.6 give a weight of 0.503 with a low frequency of 430 cps. After 13 iterations, a weight of 0.469 is obtained with a low frequency of 558 cps. The design variables final values are thickness fractions of 0.45, 0.35, 0.20; angles of 90, 90, 45 degrees; and fiber volume ratios of 0.6, 0.6, 0.6.

2.3.1.5 Cost Tailoring

The object function is cost, which is minimized. The test case consisted of a cantilever beam modeled as eight 8-noded bricks. Each brick consists of 3 layers of composite materials. The material angles, percent of thickness, and primary fiber volume ratios for each of the 3 layers were the 9 design variables used. A constraint of the lowest natural frequency was also used. The cost per unit volume of the primary fiber was input.

The 3 composites and the normalized costs used were:

AS--	@ 5.6 cost per unit fiber volume
T300	@ 2.3 cost per unit fiber volume
SGLA	@ 1.1 cost per unit fiber volume

Cost of the layup started at 13.2 with ply angles of 90, 90, and 45 degrees; ply thickness fractions of 0.30, 0.25, and 0.45; and fiber ratios of 0.6, 0.6, and 0.6. After 14 iterations cost of the layup was reduced to 9.6. The ply thickness fractions changed to 0.14, 0.22, and 0.64, respectively. The other design variables remained the same.

2.3.1.6 Thermal Tailoring

Two different thermal tailoring procedures were set up. In the first case the maximum nodal temperature is minimized. The model used consists of a 1 inch square plate made of 4 layers of a composite material, containing 96 nodes and 9 20-noded elements. There are 10 pound forces at 2 nodes and the nodes along an edge are fixed. All nodes are initially at 100°F. The heat transfer input specifies that one face of an element is cooled by convection and there is a heat flux of 12960 btu/hr on the face of another element. Four nodes have prescribed (fixed) temperatures of 100°F. The optimization input specifies that the fiber angles, percent of thickness, and fiber volume ratios of

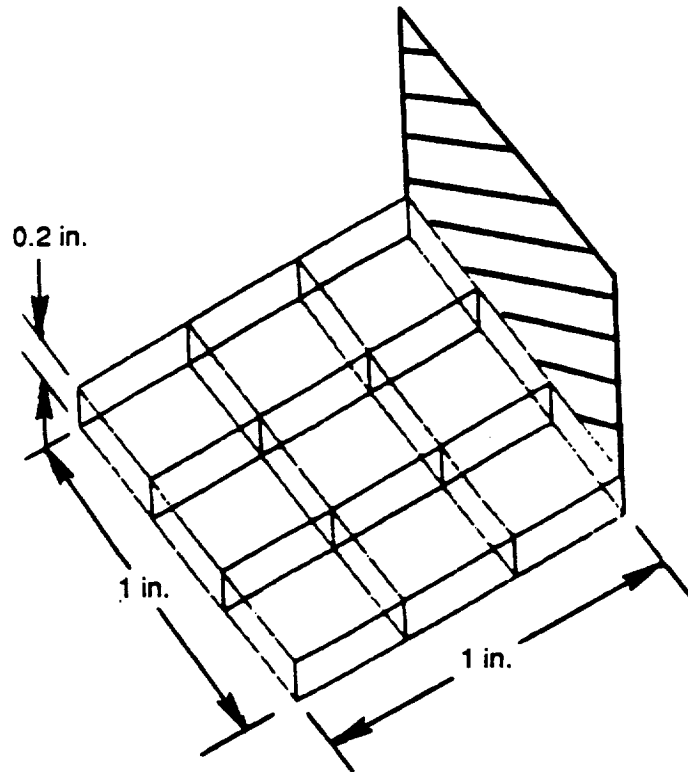


Figure 48. Frequency Tailoring Model.

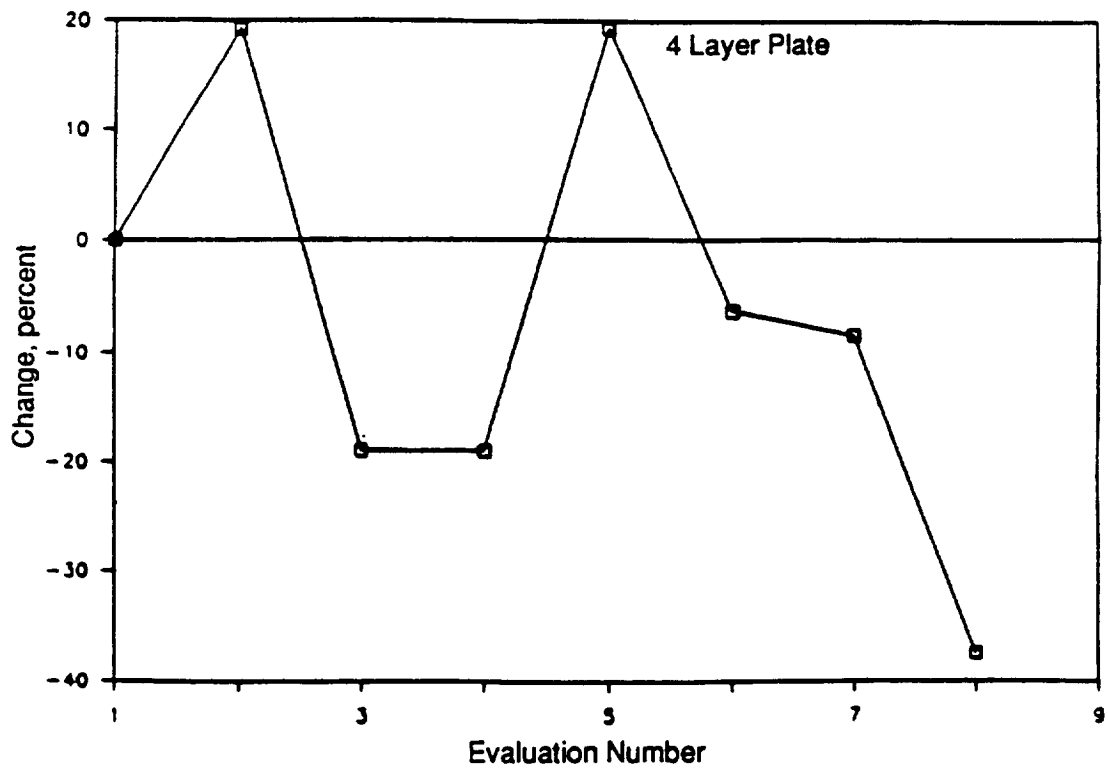


Figure 49. Frequency Tailoring Results.

the 4 layers are design variables and the maximum nodal temperature is the object function (to be minimized). Initial angles of 90, 90, 45, 45 degrees; thickness fractions of 0.30, 0.25, 0.30, 0.15; fiber ratios of 0.6, 0.6, 0.6, 0.6 give a maximum temperature of 124°F. After 16 iterations the design variables have changed to angles of 86, 45, 43, 44 degrees; thickness fractions of 0.28, 0.24, 0.27, 0.20; and fiber volume ratios of 0.6, 0.6, 0.6, 0.6. The final maximum temperature is 119°F.

The second heat transfer tailoring procedure minimizes a temperature gradient. The finite element model used is a cantilever beam 8 inches long and 3 inches thick consisting of 20 8-noded brick elements and 102 nodes. Each element is composed of 3 layers of composite materials. The fixed end of the beam has a prescribed temperature of 1000°F. The free end is cooled by convection. The optimization design variables are the same as for Case 1. The object function is the difference between the maximum and minimum nodal temperatures and is to be minimized. Initial thickness fractions of 0.33, 0.33, 0.34; angles of 45, 30, 90 degrees; fiber ratios of 0.6, 0.6, 0.6 give a temperature gradient of 490 degrees. After 280 iterations, the thickness fractions are 0.67, 0.31, 0.02; angles 90, 0, 90 degrees; fiber ratios of 0.6, 0.6, 0.6 give a temperature gradient of 309°F.

2.3.2 Simulated Components

2.3.2.1 Compressor Blade

A simulated composite compressor blade was used to demonstrate the tailoring of mode shape slopes, a criteria used in stability analysis of blades. The blade model consists of 100 8 node brick elements and 242 nodes. It is based on the J85 Stage 5 compressor blade, which is made of titanium. In order to change the vibration characteristics of the blade, the simulated model consisted of 4 simulated composite materials distributed through the blade thickness using the automatic layering capability of CSTEM. The generation set consisted of 8 plies of the 4 different materials. Figure 50 shows the blade model.

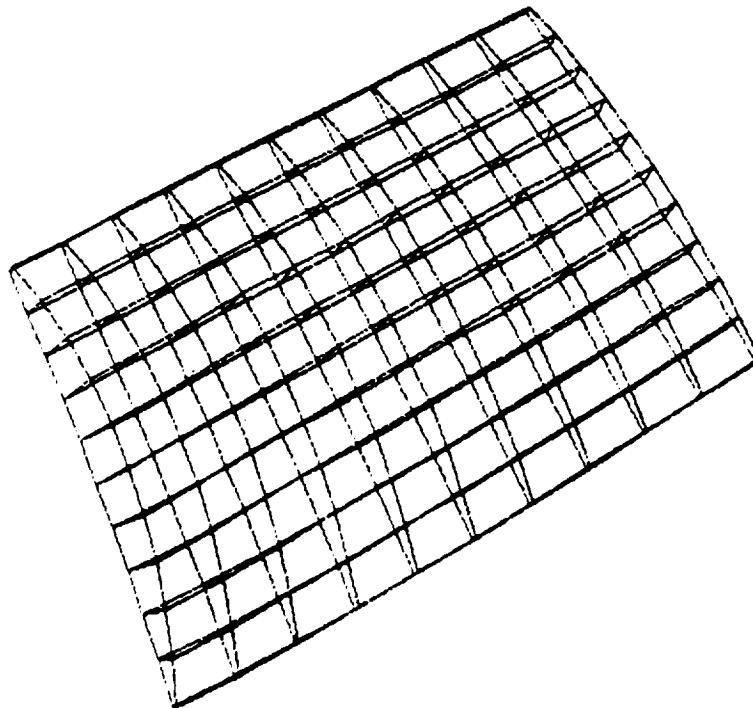


Figure 50. Compressor Blade Model.

The thickness of the generation set layers and their orientation angles comprised the 16 design variables, with the object function being the maximum mode shape slope of the lowest natural frequency. After 42 iterations and 488 CPU seconds on the CRAY-XMP this slope was reduced from 0.316 to 0.254. The gradient vector showed the thicknesses have very little effect on this slope, while the greatest effect came from the orientation angles.

2.3.2.2 Fan Blade

The fan blade shown in Figure 51 was tailored to minimize weight while keeping the lowest natural frequency above a value of 20 cps. The layering is done using the automatic layering capability of CSTEM. Three separate generation sets are used. Each generation set contains a separate material. The first set contains 6 layers of equal thickness composite material in a sequence of orientation angles. The density of this material is 0.055 lb/in^3 and an initial layer thickness of 0.016 inches. The second set contains 4 layers of equal thickness composite material in an orientation sequence with a density of 0.056 lb/in^3 and initial thickness of 0.030 inches. The material of the third set is titanium having a density of 0.161 lb/in^3 and is used to fill the model cross section thickness remaining after the first 2 generation sets have been used. The initial amount of the cross section thickness filled by the first two sets is 0.36 inches in the area where the titanium core occurs.

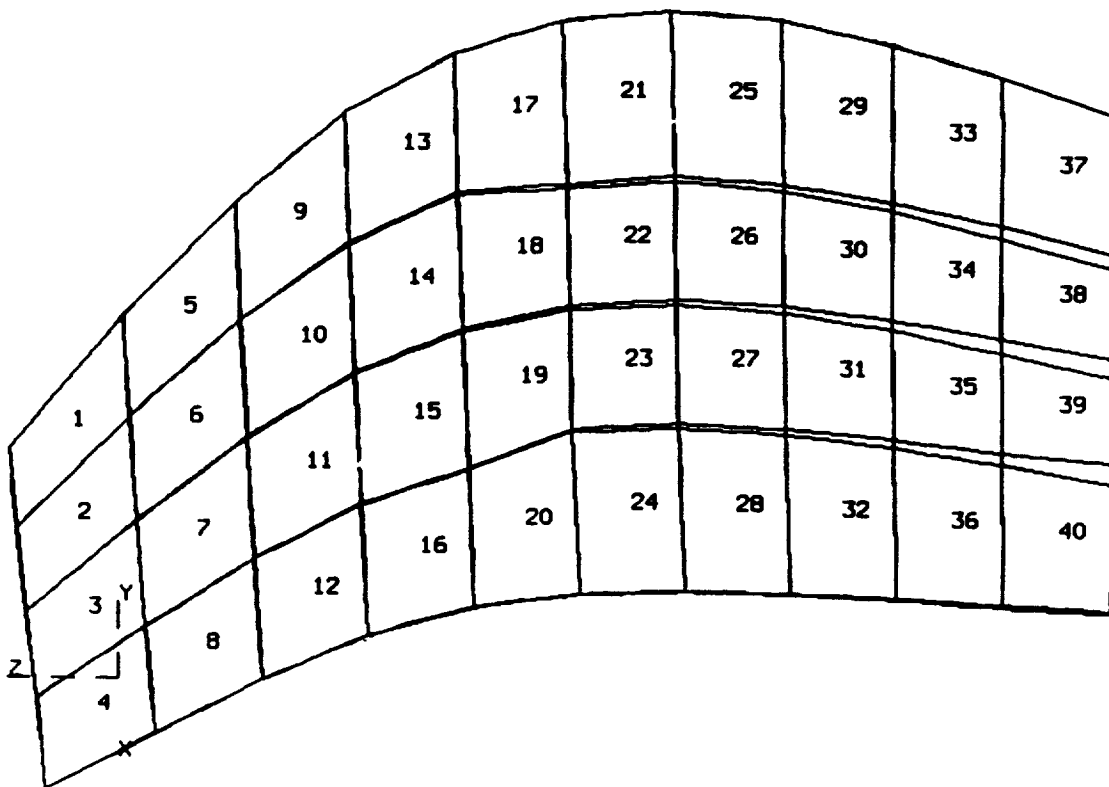


Figure 51. Coarse Mesh Composite Fan Blade.

Figure 52 shows the change of the blade weight and lowest vibration frequency throughout the tailoring process. The change in blade weight is expressed as percent of the original weight of 26.2 lbs. The final blade weight is 17.7 lbs. The frequency is expressed as percent above the constraint value. The initial frequency is 32.5 cps and the final is 21.0 cps. Figure 53 shows the layup of the element with the thickest cross section at several evaluation points. The thickness of this element is 1.868 inches. It can be seen that the tailored blade contains only composite material and that the amount of Material 1 has been increased to a thickness fraction of 0.08026. This corresponds to a thickness of 0.15 inches, which is the allowed upper bound input to the tailoring procedure.

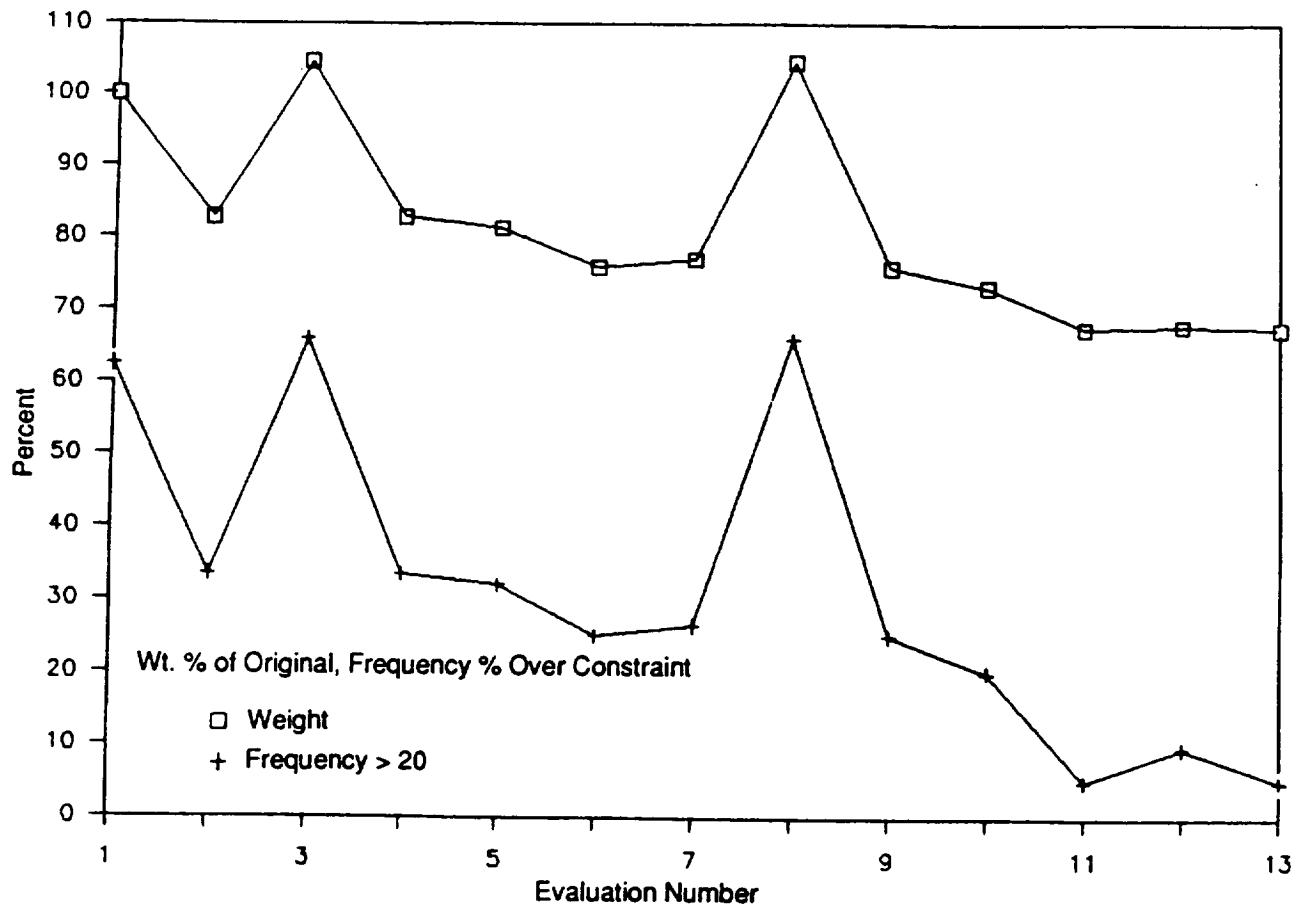


Figure 52. Weight Tailoring of Fan Blade.

Layer	Matl	Fraction	IAX	Angle
1	1	0.8562E-02	3	125.0
2	1	0.8562E-02	3	215.0
3	1	0.8562E-02	3	170.0
4	1	0.8562E-02	3	215.0
5	1	0.8562E-02	3	125.0
6	1	0.8562E-02	3	80.00
7	2	0.1605E-01	3	125.0
8	2	0.1605E-01	3	170.0
9	2	0.1605E-01	3	125.0
10	2	0.1605E-01	3	80.00
11	3	0.1605E-01	3	0.0000
12	3	0.1605E-01	3	0.0000
13	3	0.1605E-01	3	0.0000
14	3	0.1605E-01	3	0.0000
15	3	0.1605E-01	3	0.0000
16	3	0.1605E-01	3	0.0000
17	3	0.1605E-01	3	0.0000
18	3	0.1605E-01	3	0.0000
19	3	0.1605E-01	3	0.0000
20	3	0.1605E-01	3	0.0000
21	3	0.1605E-01	3	0.0000
22	3	0.1605E-01	3	0.0000
23	3	0.1605E-01	3	0.0000
24	3	0.1605E-01	3	0.0000
25	3	0.1605E-01	3	0.0000
26	3	0.1605E-01	3	0.0000
27	3	0.1605E-01	3	0.0000
28	3	0.1605E-01	3	0.0000
29	3	0.1605E-01	3	0.0000
30	3	0.1605E-01	3	0.0000
31	3	0.1605E-01	3	0.0000
32	3	0.1605E-01	3	0.0000
33	3	0.1605E-01	3	0.0000
34	3	0.1605E-01	3	0.0000
35	3	0.2362E-20	3	0.0000

Evaluation 1
(Original)

Layer	Matl	Fraction	IAX	Angle
1	1	0.4013E-01	3	125.0
2	1	0.4013E-01	3	215.0
3	1	0.4013E-01	3	170.0
4	1	0.4013E-01	3	215.0
5	1	0.4013E-01	3	125.0
6	1	0.4013E-01	3	80.00
7	2	0.1605E-01	3	125.0
8	2	0.1605E-01	3	170.0
9	2	0.1605E-01	3	125.0
10	2	0.1605E-01	3	80.00
11	3	0.1605E-01	3	0.0000
12	3	0.1605E-01	3	0.0000
13	3	0.1605E-01	3	0.0000
14	3	0.1605E-01	3	0.0000
15	3	0.1605E-01	3	0.0000
16	3	0.1605E-01	3	0.0000
17	3	0.1605E-01	3	0.0000
18	3	0.1605E-01	3	0.0000
19	3	0.1605E-01	3	0.0000
20	3	0.1605E-01	3	0.0000
21	3	0.1605E-01	3	0.0000
22	3	0.1605E-01	3	0.0000
23	3	0.5572E-02	3	0.0000

Evaluation 6

Layer	Matl	Fraction	IAX	Angle
1	1	0.7100E-01	3	125.0
2	1	0.7100E-01	3	215.0
3	1	0.7100E-01	3	170.0
4	1	0.7100E-01	3	215.0
5	1	0.7100E-01	3	125.0
6	1	0.7100E-01	3	80.00
7	2	0.1605E-01	3	125.0
8	2	0.1605E-01	3	170.0
9	2	0.1605E-01	3	125.0
10	2	0.1284E-01	3	80.00
11	3	0.1299E-01	3	0.0000

Evaluation 11

Layer	Matl	Fraction	IAX	Angle
1	1	0.5351E-02	3	125.0
2	1	0.5351E-02	3	215.0
3	1	0.5351E-02	3	170.0
4	1	0.5351E-02	3	215.0
5	1	0.5351E-02	3	125.0
6	1	0.5351E-02	3	80.00
7	2	0.1605E-01	3	125.0
8	2	0.1605E-01	3	170.0
9	2	0.1605E-01	3	125.0
10	2	0.1605E-01	3	80.00
11	3	0.1605E-01	3	0.0000
12	3	0.1605E-01	3	0.0000
13	3	0.1605E-01	3	0.0000
14	3	0.1605E-01	3	0.0000
15	3	0.1605E-01	3	0.0000
16	3	0.1605E-01	3	0.0000
17	3	0.1605E-01	3	0.0000
18	3	0.1605E-01	3	0.0000
19	3	0.1605E-01	3	0.0000
20	3	0.1605E-01	3	0.0000
21	3	0.1605E-01	3	0.0000
22	3	0.1605E-01	3	0.0000
23	3	0.1605E-01	3	0.0000
24	3	0.1605E-01	3	0.0000
25	3	0.1605E-01	3	0.0000
26	3	0.1605E-01	3	0.0000
27	3	0.1605E-01	3	0.0000
28	3	0.1605E-01	3	0.0000
29	3	0.1605E-01	3	0.0000
30	3	0.1605E-01	3	0.0000
31	3	0.1605E-01	3	0.0000
32	3	0.1605E-01	3	0.0000
33	3	0.1605E-01	3	0.0000
34	3	0.1605E-01	3	0.0000
35	3	0.2362E-20	3	0.0000
36	3	0.5572E-02	3	0.0000

Evaluation 3

Layer	Matl	Fraction	IAX	Angle
1	1	0.4883E-01	3	125.0
2	1	0.4883E-01	3	215.0
3	1	0.4883E-01	3	170.0
4	1	0.4883E-01	3	215.0
5	1	0.4883E-01	3	125.0
6	1	0.4883E-01	3	80.00
7	2	0.1605E-01	3	125.0
8	2	0.1605E-01	3	170.0
9	2	0.1605E-01	3	125.0
10	2	0.1605E-01	3	80.00
11	3	0.1605E-01	3	0.0000
12	3	0.1605E-01	3	0.0000
13	3	0.1605E-01	3	0.0000
14	3	0.1605E-01	3	0.0000
15	3	0.1605E-01	3	0.0000
16	3	0.1605E-01	3	0.0000
17	3	0.1605E-01	3	0.0000
18	3	0.1605E-01	3	0.0000
19	3	0.1605E-01	3	0.0000

Evaluation 10

Layer	Matl	Fraction	IAX	Angle
1	1	0.8026E-01	3	125.0
2	1	0.8026E-01	3	215.0
3	1	0.8026E-01	3	170.0
4	1	0.8026E-01	3	215.0
5	1	0.8026E-01	3	125.0
6	1	0.8026E-01	3	80.00
7	2	0.1605E-01	3	125.0
8	2	0.2362E-02	3	170.0

Evaluation 13

Figure 53. Tailored 1/2 Symmetric Layups.

3.0 CONCLUSIONS

The tasks in the NASA Statement of Work for the CSTEM program have been successfully completed. A finite element computer program that has the capability to analyze the structural, thermal, electromagnetic, and acoustic characteristics of graded composite materials has been developed and demonstrated on various test cases and simulated components. Included in this program is the capability to tailor structures so that a particular characteristic is enhanced while satisfying other constraints. This was also demonstrated successfully. In addition, the CSTEM computer program has been written in a modular form so that it should be relatively easy and convenient to add to and modify the features of the program.

The capability exists in the CSTEM computer program to perform comprehensive analyses of numerous material types ranging from isotropic to advanced graded composites. The tailoring of these structures using the coupled analyses contained in the CSTEM program is a unique and exciting capability which may require some experience in order to take full advantage of all aspects of the program. The possibility of adding to and enhancing these capabilities is another interesting and potentially beneficial prospect.

REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-0188	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.				
1. AGENCY USE ONLY (Leave blank)		2. REPORT DATE April 1992		3. REPORT TYPE AND DATES COVERED Final Contractor Report
4. TITLE AND SUBTITLE Coupled Structural/Thermal/Electromagnetic Analysis/Tailoring of Graded Composite Structures Final Status Report			5. FUNDING NUMBERS WU-505-62-91 C-NAS3-24538	
6. AUTHOR(S) M.S. Hartle, R.L. McKnight, H. Huang, and R. Holt				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) General Electric Aircraft Engines 1 Neumann Way Cincinnati, Ohio 44135			8. PERFORMING ORGANIZATION REPORT NUMBER None	
9. SPONSORING/MONITORING AGENCY NAMES(S) AND ADDRESS(ES) National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135-3191			10. SPONSORING/MONITORING AGENCY REPORT NUMBER NASA CR-189153	
11. SUPPLEMENTARY NOTES Project Manager, C.C. Chamis, Structures Division, NASA Lewis Research Center, (216) 433-3252.				
12a. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified - Unlimited Subject Category 39			12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words) Accomplishments are described for the 5-year program to develop a methodology for coupled structural/thermal/electromagnetic analysis/tailoring of graded component structures. The capabilities developed over the term of the program are the analyzer module and a tailoring module for modeling of graded materials are summarized. Highlighted accomplishments for the past year include: (1) addition of buckling analysis capability; (2) addition of mode shape slope calculation for flutter analysis; (3) verification of the analysis modules using simulated components; (4) verification of the tailoring module.				
14. SUBJECT TERMS FEM structural analysis; FEM thermal analysis; FEM electromagnetic analysis; Graded composites; Coupled analysis; Optimization; Tailoring			15. NUMBER OF PAGES 54	
			16. PRICE CODE A04	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT	